

Mirrored Parts

There are several methods available for modeling mirror-image parts in SolidWorks. The selection of a technique is dependent on how you want changes to propagate. Should changes to the mirrored part be editable and automatically propagated between similar parts? Or should your mirrored part be independent of editing changes to the original part? Also, how will these part files be treated in your company document control system?

- **SolidWorks**
- *COSMOS*
- *PDM*
- *Rapid Prototyping*
- *Office Productivity Tools*

There are three approaches to modeling mirror-image parts in SolidWorks. The specific functionality I will discuss is covered in the standard SolidWorks training material in three places: the Essentials class; the Advanced Assembly class; and the Advanced Shapes class. This discussion takes you beyond the button pushing of creating a mirrored part and delves into the merits and consequences of each approach.

About KAP

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Keith Pedersen has a BSME from Clarkson College, and an MSME from Boston University. After a stint at General Electric in Burlington, VT, Keith was the lead Applications Engineer for Advanced Surfacing products for Matra Datavision USA, including EUCLID-IS, UniSurf, and STRIM. He joined CAP in 1998 to support advanced surfacing applications in SDRC I-DEAS and joined our SolidWorks group one year later. Keith has extensive industry and consulting experience in non-linear Finite Element Analysis and Computational Fluid Dynamics in addition to surfacing applications. He is a Certified SolidWorks Professional (CSWP) and certified to train and support COSMOSWorks.

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Method 1: Save the mirror image in a new (derived) part file.

This is the most automatic method, and the approach that is emphasized in the training material. There are two ways to achieve this kind of mirroring, depending on whether the file you are starting from is a Part or an Assembly.

KAP's Tip: Pre-select a face or plane to activate menu items needed for inserting mirrored parts.

From a Part file: The menu INSERT – MIRROR PART does the job. This menu item is usually grayed-out, which would seem to indicate it is unavailable, but the trick here is that you must pre-select a face or plane to mirror the part about, before this menu item is activated. This will result in the creation of a new Part file, which will contain a single feature, with a name like this;

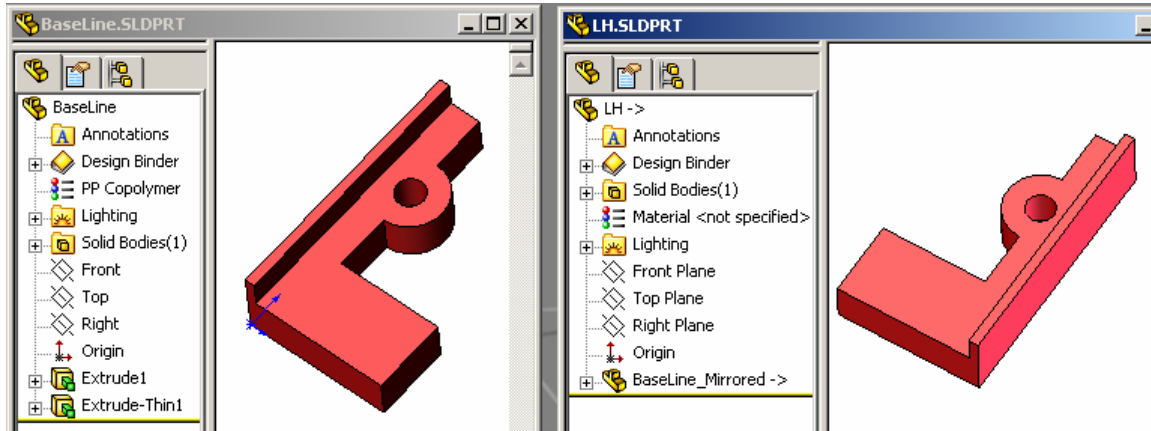
partname_Mirrored ->

The right-arrow symbol after the name indicates that this feature is a File reference, and so if the original RH (right-hand) part file is ever modified, the new LH (left-hand) file will update to show the changes also.



KAP's Tip: Place a check mark in the wizard for all the parts you want mirrored. Unchecked parts will be instantiated in their original position.

From an Assembly file: The menu INSERT – MIRROR COMPONENTS produces the same results I've outlined above, except that it is a wizard-based procedure that automates the process for any number of parts in the assembly at a single go. For each part that you identify in the first selection list, you have a check-box to indicate that you would like an opposite-handed part file to be spawned. (If you don't check the box for a particular part, it will not spawn a LH version of the part file, but simply position a new instance of the original part in the mirrored location).



Baseline Part File has the Feature History

LH Part file has a single Reference Feature

Advantages

The beauty of this approach is the automatic updating. Also, if your company uses a document control system, (PDM, MRP, etc) that requires a unique CAD file reference for each Control Number, then this is the way to do it. You can edit the Custom File Properties of the left-hand version independently of the original, so it will be easier to properly attribute the LH and RH versions in the Bill of Materials.

If the LH and RH parts are not perfect mirror-images, but have some features unique to each, the best way to model them may not be using two part files, but with three! Let's assume that you build a RH version of the part, up to the level of detail that is common to both. Save this file off as the Baseline part file. Then, you use the MIRROR PART command to spawn a second part file, and here you construct any additional features that are unique to the LH. Finally, you open a new, empty part file, and use the command INSERT - PART to pull in a file reference of the Baseline part. You then place any features unique to the RH in this file. In either case, MIRROR PART and INSERT PART create a single feature with a file reference back to the baseline. This direct referencing of a part-to-part is sometimes referred to in the SolidWorks help files as a "Derived Part", and in general engineering jargon we call this a "Make-From" situation.

KAP's Tip: Understanding the subtle differences in these command modalities of the SolidWorks interface can prevent confusion.

KAP's Tip: Using linked driving dimensions can result in ambiguity when using drawings to document mirrored parts created via this derived technique.

It is a peculiarity of the SolidWorks interface that you use the MIRROR PART command to “push” the file reference, but that with INSERT PART you must “pull” the file reference. These functions are found adjacent to each other in the same menu. They do practically the same job, and produce the same kind of external references. Yet you don't invoke them the same way.

For the MIRROR, you must first open the Baseline file, and from within this file you 'push' the reference out to create the LH file.

For the INSERT PART function, you do NOT open the Baseline file. Instead, you open a new, empty part file, and then use the INSERT command to browse to the Baseline file and 'pull' the reference in.

Disadvantages

None of the sketches or features dimensions of the baseline part are available to the user in the Mirrored/Derived result. Except for the fact that the LH part will automatically update with respect to any changes, it otherwise feels to the user as if it were imported from IGES or STEP, etc. This makes it difficult to create drawings using Driving (linked) dimensions.

If you want to make a separate drawing for the LH part, you could envision solving this problem by having two (or more) views; One drawing view is of the Mirrored part, showing the dimensions of the features unique to the LH. The other view would have to be of the Baseline part file, showing the dimensions of the features common to both LH or RH. Many users dislike the aesthetics of having one or more RH views in a drawing that is supposed to document the LH part, and fear that confusion could result. The fall-back position, then, is to simply advise users to never use Driving dimensions to document a mirrored part. Any mirrored features must be dimensioned manually, or via the AutoDimension, but not by using INSERT – MODEL ITEMS.



Method 2: Save the mirror image in a new configuration.

This method uses the power of Configurations, plus the ability to create and act upon multiple solid bodies within a Part file. The procedure is straightforward. First, build the part up to the level of detail that is common to both the LH and RH designs. You can leave this configuration named "Default", but I prefer to rename this configuration "BaseLine". Then create two more configurations, (which I'll call LH and RH). In the RH configuration, you simply add the features (if any) that are unique to the Right-Hand design.

In the LH configuration, you now insert an operation to mirror the entire solid body; INSERT – PATTERN/MIRROR - MIRROR. After filling in the "Bodies to Mirror" prompt, look just below it in the dialog, and un-check the option "Merge Solids". After you hit 'Go', there will be a folder in the Feature Manager showing two solid bodies, the latter one being the mirrored result. If you now right-mouse-click over the icon for the earlier of these two bodies, and select the function DELETE BODY, you will be left with only the mirrored result on the screen.

Important Note: The last sentence above does not ask you to DELETE any features. Instead, you are creating a new feature and adding it to the feature manager – and the name of that new feature operation is "Delete Body".

So – Don't delete any features ...

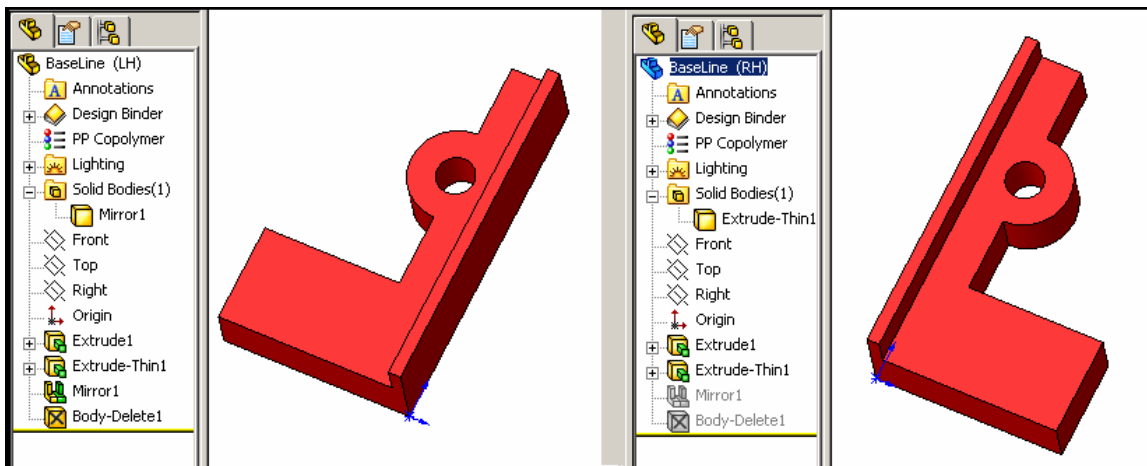
Instead - Add a feature named "Delete"...,

Confused?

The multi-body capability in SolidWorks is explained copiously in the Advanced Shapes class, but all we need to know here is that

*KAP's Tip: I explain
the semantics of the
"Delete Body"
command*

by invoking the DELETE BODY command, you tell the system that the baseline, or RH version of the body is not needed in the LH configuration, leaving you only with the mirrored result. And, the Delete Body operation is configurable. So if you activate the Baseline or RH configurations now, you will find both the Mirror and the Delete Body features are suppressed.



Left Hand Configuration creates the mirror-image solid, and deletes the original

Advantages

There is only one CAD file to keep track of. All the sketches, dimensions, and reference geometry are locally available for reference and re-use. If these parts are to be used in an assembly, and you want them to report different properties into the Bill of Materials, you can do so by assigning Custom File Properties that are specific to each configuration.

Disadvantages

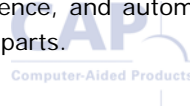
This approach improves the convenience and editability of the part file, but doesn't solve any of the issues stated above about creating drawings of the LH part. In fact, if you create a drawing



view of the LH configuration, and then use INSERT – MODEL ITEMS to get the feature dimensions, you'll get a whole lot of dimensions hanging out in space beside the part – where the right-hand version used to be, before that body was deleted. So once again, you fall back on limiting yourself to using Driven dimensions. Similarly, if you were to double-click on any feature early on in the Feature Manager, you'll see the dimensions appear – in the RH position. This is because going back to edit an earlier feature is essentially going back in time – and prior to creating the MIRROR BODY feature, the part really is right-handed.

Method 3: Re-create the opposite-hand part in a new file.

There, I said it. This is not a fun thing to suggest. But, every now and then, I encounter a design application where the need for local, independently editable features, and the need for linked, Driving dimensions in the drawing, outweigh the desire for speed, convenience, and automatic updating of changes between the LH and RH parts.



I find that one seldom has to build the LH part completely from scratch, however. First, you use FILE – SAVE AS to save the RH part off with the LH file name. Then, roll the Time-Bar of the feature manager up to the first feature in the tree. Edit the features one-at-a-time, either reversing the extrude or revolve directions, or using the MODIFY SKETCH tool to re-orient the sketches, (this command allows you to mirror, flip, and rotate sketches, as long as you have first removed any external references). As you roll the Time-Bar down thru the tree, you will doubtlessly encounter broken and confused features, but it is still going to be faster to build your new features over the bones of the old one, compared to starting out again from scratch.

KAP's Tip: Use COPY and PASTE to remove external file references from a sketch.

In these cases, I'll work with the RH and LH part windows tiled side-by-side, so I can make use of the ability to COPY and PASTE sketches from one to the other. This helps me keep track of what



feature I'm working on, and when you PASTE in a sketch from another part, it automatically strips out any of the external references, so you're all ready to flip the sketch orientation without having to manually hunt down and remove external dimensions.

KAP's Tip: The good...

Advantages

The resulting LH part will be completely independent, free of any references to other files. Any roll-back edits and parameter-edits will display dimensions and call-outs that are spatially Left-Handed and so display properly on the model. There is no need to create additional Configurations. All dimensions can be imported into the LH drawing as Driving dimensions.

KAP's Tip: ...the bad, and the ugly...

Disadvantages

Ugh. It's a lot of work.

No automatic change propagation. Any changes to the RH model will have no effect on the LH model, so you'll have to manually reproduce any symmetric edits.

CONCLUSION

Most of the points I've made here can also apply to the creation of a family of "Make From" parts, where a common casting or forging might be pulled from stock, then custom-machined to produce a variety of different finished products.

The most important consideration is to assess which is more important – independence, or automatic propagation of changes between the similar parts. Understanding the ramifications of each of these methods of creating a mirrored part can minimize the downstream updating, maintenance and documentation of your parts and assemblies.