AUTODESK UNIVERSITY 2005
Walt Disney World Swan and Dolphin Resort
Orlando, Florida

Derived Parts in Autodesk Inventor®

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MA15-1L Derived parts are a powerful but easy-to-use too in Inventor. They come in two basic flavors: you can derive a part from another one, or you can derive a part from an assembly. Using derived parts, you can easily create machining models and drawings from an as-cast model, and you can create a mold from the same model. The mold cavity can be scaled to allow for shrinkage. Opposite-hand versions can also be created from the base part and from any subsequent derived parts. If the basic part is changed then everything updates.

About the Speaker:

An AutoCAD software user since 1986, Bill was a product engineer and manager for Weiser Lock in Vancouver, Canada for 27 years. Bill has taught AutoCAD and mechanical design at the British Columbia Institute of Technology since 1996 and teaches Autodesk Inventor at the Institute’s Training Center. He has lectured on a wide range of subjects at Autodesk University since 1995. An active member of the Vancouver AutoCAD Users Society, he has written "The Learning Curve" column for CADalyst magazine since 1986, and writes about Autodesk Mechanical Desktop and Autodesk Inventor for Autodesk's Point A Toplines. He also writes for Inside AutoCAD Journal and Design Product News.

NOTE

This lab is intended for INTERMEDIATE users. I assume you are familiar with the basic operations within Inventor, including the production of parts, assemblies, and 2D working drawings.

The Deriving Force…

Most of the time, the production of a part model in Inventor is a serial process. You start with a single sketch, dimension and constrain it, then turn it into a solid. The process is repeated, adding and subtracting features, until the part is finished. The Browser shows this sequence. Yes, under the right conditions you can drag individual features up and down the browser to change their relative position within the history tree, but the final result is still a sequential process.

This sequential process serves us well enough most of the time, but there are two main situations in which it can be a problem:

1. A primary part such as a forging receives secondary machining operations. You need both a forging drawing and a machining drawing.
2. You need a part that is actually a negative image of another; you have modeled a complex part and now you need the dies or molds for it.

The Party Of The First Part…

Let's start with the first case, where you need a primary drawing and a machining drawing. The brute force method would be to model the primary part and then the machining features, and then create the 2D drawing.

Now, when you want the primary part drawing you suppress the machining details and plot the drawing. To get the machining drawing, unsuppress the machining details in the part, and then suppress all of the primary part dimensions in the drawing… hey, this could get out of hand!

It gets worse. What if the same primary part branches into two or more different machined configurations, or what if it goes through several levels of secondary operations? What if you want the primary part and the machining details in one 2D drawing?

Okay, now let's look at the second case. Having built the part, how do you create the mold or die for it? The brute force method would be to build the part from scratch again, but this time you start with a solid block and then reverse all the Boolean operations; subtract instead of add and vise-versa. Meanwhile, you have to allow for shrinkage as the part solidifies and cools, and by the way we also need a left-hand version of the part…

This Is So Derivative...

Fortunately there is a better way of solving both situations. Actually, it is doubly fortunate. Not only do we get a better way of doing things, but we also get a good tutorial topic.

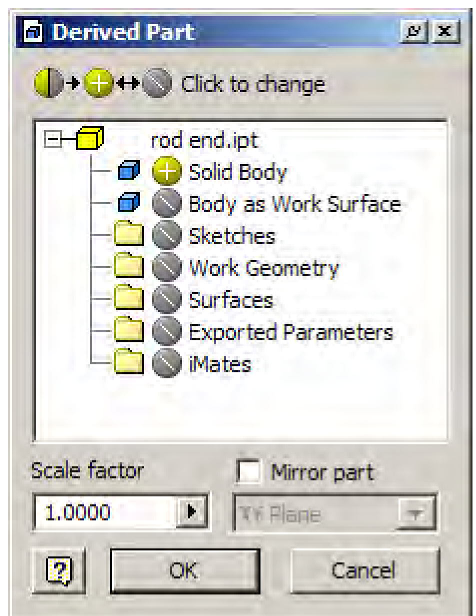
The better way is called "derived parts". Basically, this means that we can create a part or an assembly and then derive other parts from it. Parametric relationships are still maintained, so that if the base part changes, or if a part within an assembly changes, then the derived part or parts will follow.

The End Is Near…

The first case, where we derive a machined part from a primary one, is a little simpler so we will start with it.

1. Activate the **C:\datasets\MA15-1L\Derived parts.ipj** project.
2. Open the file <path>**Rod end.ipt**. This step is not essential to the process, but it will help you to see what we are doing.
3. Start a new part from the **English \ Standard(in).ipt** template
4. Click **Return** to cancel the sketch.
5. Click **Derived Component** down near the bottom of the Part Features panel.
6. In the file dialog, select and open the **Rod End.ipt** file

7. The Derived Part dialog box appears. If you click on a grey round \ button then it will turn into a yellow + one. For our purposes at this time the only yellow + button should be the Solid Body one. We will come back later to investigate what the other ones do.



8. Click OK. The part is inserted into the new file.

   At first glance this file would appear to be exactly identical to the *Rod end.ipt* file opened earlier, but a closer examination reveals that the browser is different. The original file shows the full structure of all the features in the part, whereas the new file simply contains the rod end as a derived part.

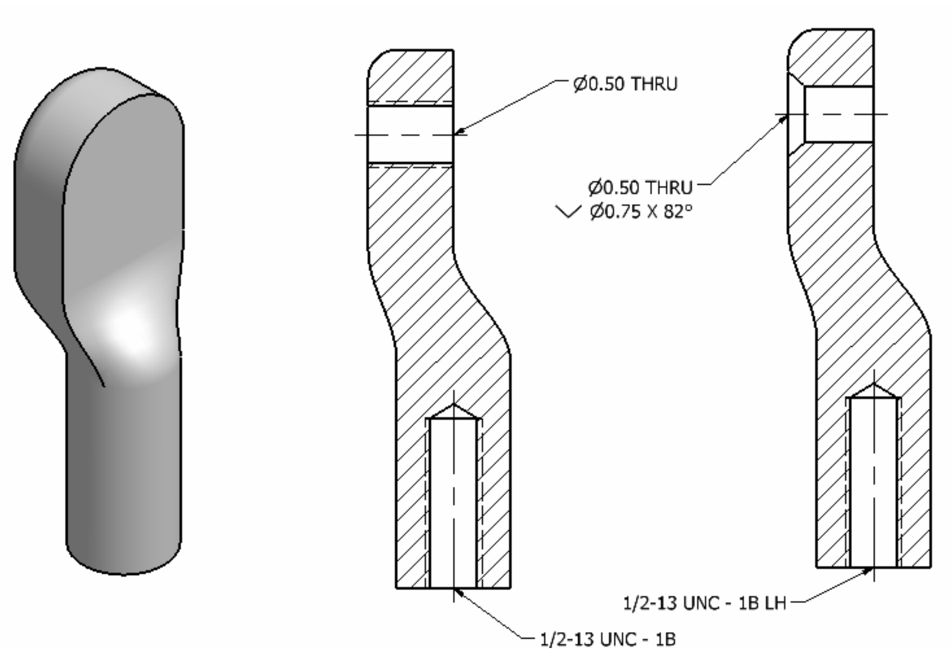
   AutoCAD users would call this an XREF; the new file contains an external reference link back to the first part.

   You can now add, subtract, or intersect any new features, just as if you were still working on the original part. Let's go ahead and add some machining details.
9. Place a ½-13 UNC ANSI Unified thread 1-1/2" deep in the bottom end.
10. Place a ½-13 UNC, ANSI Unified thread 1-1/2" through the upper tab.
11. Save the file as **A-U Rod end – threaded.ipt**
12. Repeat steps 3 –9, but make it a left-hand thread in the bottom end.
13. Place a ½" hole with a .750" countersink through the upper tab.
14. Save the file as **A-U Rod end – c'sink.ipt**

The following illustration was taken from a single drawing file that references all three of the part files. The two machined parts are actually children of the base part and are dependent on it. When the base part is changed the two machined parts will update. Let's try that.

15. Open the 2D drawing file **Rod End.idw.**

16. Click **Window | Arrange All** to show all four files at once.



17. Click the base Rod End window to make it current.

18. In the browser, double-click Extrusion2 and change the 4.00 length to 5.00. Click on Update.

19. Click in a machined part window to activate it, and then click on Update. Repeat for the other machined part.

20. Click in the drawing window. All three parts will update to show the change made to the base part. Magic!

If It Ain't Broken…

There may be situations wherein you want to break the link between the derived part and its base part. File management might be a reason for doing this. If you want to send someone else a copy of the derived part you either need to send the base part as well, or you need to break the link to the base part. This is equivalent to binding an XREF in AutoCAD. The definition of the base part within the derived part gets frozen at its current value, and it cannot be edited within the derived part file.

To accomplish this, all you need to do is to go to the browser for the derived part and right-click on the entry for the base part. Now just click on the Break Link… item in the context menu that appears. Done!

The Optional Extras…

Earlier, we mentioned the options in the Derived part dialog box. Let's look at the others.

Usually, the default is to have a single yellow + button as shown earlier. This means that the base part gets attached as a solid part. The second item in the dialog box attaches the part as a surface model. These two are radio buttons; you can have one or the other, but not both. You can have neither, but this is only meaningful if one or more of the other 5 buttons is yellow + and conditions are right.

"Conditions are right" means the other objects are available in the base part. For example, if the base part has an unconsumed sketch and you turn this option on whilst creating the derived part then the sketch will be available in the derived part for use in subsequent operations within the derived part.

The same is true for work geometry, surfaces, and iMates. If you want to use parameters from the base part within the derived part then you must first open the base part and flag the desired parameters for export.

You can also set a scale factor between the base part and the derived part. As we will see, this can be useful for taking care of shrinkage in a cast or molded part.

The final option lets you create a derived part that is a mirror image of the base part.

As we have seen, derived parts let us create machined variants from a primary part, including multiple machined configurations starting from a common base part.

Another use is in the creation of several parts that have some features in common. For example, two parts might be 90% identical except that one has a slot and one has a tab. Simply create the base part and then derive the two variants.

And now a final fascinating fact; a derived part can in turn be the base for a derived part. In our previous example, the slotted and tabbed parts can be the bases for several machined parts.

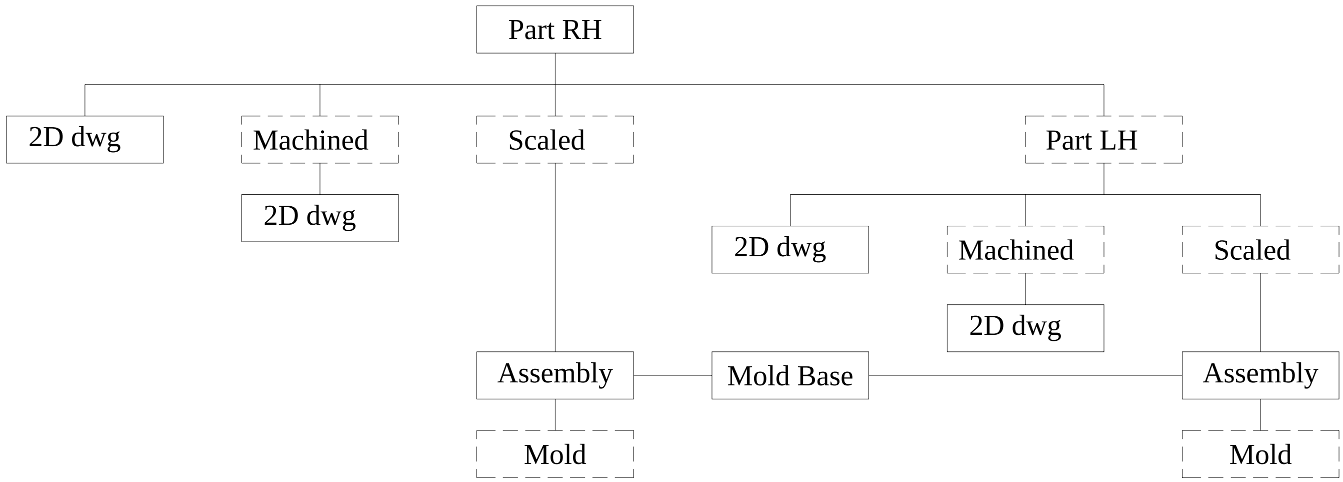
Let's Rock!

We now come to the second case, wherein we want to sink the mold cavity for a part. Actually, this is just an example case of the more generic situation wherein we derive a single part from an assembly.

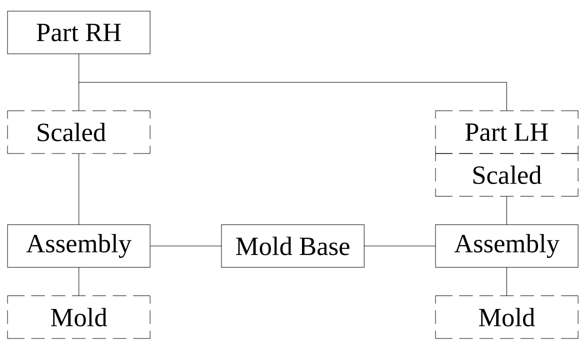
Obviously, we need an assembly before we can derive a part from it. Before Inventor added direct support for weldments, users would build the individual sections, and then assemble them into a weldment. They would then derive a single part from the assembly so they could go on to show the post-welding machining operations.

For our exercise, we are going to start with a model of the right-hand version of a machine rocker link, which is to be made from die-cast aluminum. You need to produce one half of the mold for this part, and another mold half for the left-hand part. You need to allow for shrinkage.

The following flow chart illustrates the process for a full situation in which you need 2D working drawings for each part as cast, and for the machined variants of them, and for the two mold halves. The boxes with dashed borders indicate derived parts.



We want to concentrate on the deriving process, so we will use the following simplified process chart. Note that it is usually a good idea to think things through and sketch such a flow chart first to make sure you do things efficiently and so you arrive at the intended result.

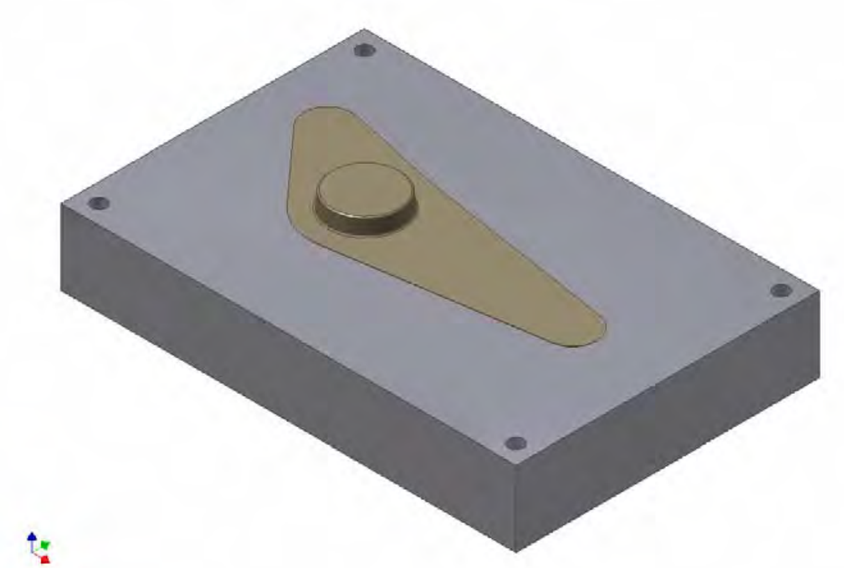


1. Open the file **Rocker-RH.ipt**. Once again, this is not essential but it helps us to understand what is going on.
2. Start a new part from the **Metric \ Standard(mm).ipt** template.
3. Click **Return** to cancel the sketch.
4. Click **Derived Component** down near the bottom of the Part Features panel.
5. In the file dialog, select and open the **Rocker-RH.ipt** file.
6. In the Derived Part dialog, set the scale factor to 1.05 and click OK. Actually, shrinkage depends on a great many factors, but 5% is a good nominal value to start from for aluminum. Your new part looks exactly like the base one, but it is 5% bigger.
7. Change the colour to Beige (Dark). This is done purely to make it easier to see within the assembly. If you don't like beige, pick anything else.
8. Save your file as **Rocker-RH-Scaled.ipt**
9. Start a new assembly file from the **Metric / Standard(mm).iam** template.

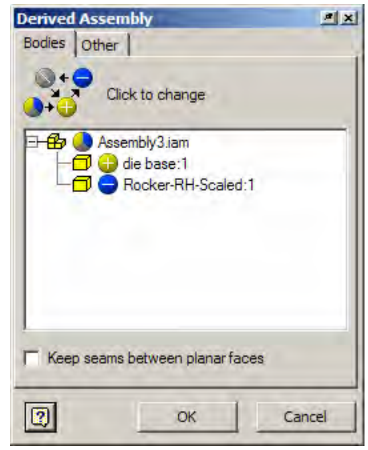
10. Place one instance of the **Die Base.ipt** component into the assembly.
11. Place one instance of your new **Rocker-RH-Scaled.ipt** component into the assembly.
12. Constrain the rocker to the die base:
    a. As a minimum, you need a Flush constraint between the top face of the die base and the underside (the one with the shorter protrusion) of the rocker.
    b. Position, and preferably constrain, the rocker approximately in the centre of the face of the die base. Your assembly should look about like this:
13. Save your file as **Assembly-Mold-RH.iam.**

We now have an assembly of the die base and the scaled RH rocker part. If you were to do an interference check, Inventor would report that most of the rocker was interfering with the base.

14. Start a new part from the **Metric \ Standard(mm).ipt** template.
15. Click **Return** to cancel the sketch.
16. Click **Derived Component** down near the bottom of the Part Features panel.



17. In the file dialog, select and open your new **Assembly-Mold-RH.iam** file. This brings up the Derived Assembly dialog box. This dialog box lists all the bodies that are to be imported into the derived part. By default, they are all marked with the yellow + button to indicate that they will be added to the derived part. As you click on each button it will toggle through a blue – button which subtracts the part to a grey \ button which ignores the part and back to the yellow + button again. For our example we want to <u>add</u> the die base and <u>subtract</u> the rocker.



18. Click on OK to produce our new die cavity. You may want to right-click on the rocker entry in the browser, then click on Properties, and change the colour to something with more contrast.

    That's all there is to it! You have now worked down the left side of our flow diagram and have created a scaled cavity from a multi-featured RH part. Now, on to the other side.

19. Repeat steps 2-18 again, with a couple of minor differences:
    a. In step 6, set the scale factor to 1.05 <u>and</u> click the Mirror Part check box. Select the X-Z Plane from the drop list, and then click OK.
    b. For the rest of the steps, substitute LH for RH in each file name.
    c. You should end up with this:

That's it! You have now created the matched pair of LH and RH scaled die cavities, all from a single base part.

This has shown the basic process. In the real world you would build the two assemblies to include two die bases, one for the hot half and one for the cold half. You would then derive two die parts from each assembly; one with the hot-half base included and the cold half excluded and vise-versa.

The More Things Change…

Not unexpectedly, things can be edited all along the line:

- If you edit **Rocker-RH.ipt,** then the changes will reflect through all the related files shown in the flow diagram.
- If you right-click on the derived body in the browser of a part derived from a single part and then click on Edit…, then the Derived Part dialog appears. You can edit the scale factor, the mirroring, and the data to be brought forward. Once again, all changes carry downstream.
- If you right-click on the assembly in the browser of a part derived from an assembly and then click on Edit…, then the Derived Assembly dialog appears. You can edit the add/subtract/ignore status. Changes carry on downstream.

…and a few final random comments.

- Having derived a part from an assembly, it becomes a normal single part. You can continue adding and subtracting features such as runners, sprues, gates, ejector pin holes, cooling passages, and so on.
- The assembly from which a part is derived can contain sub-assemblies. The components in a sub-assembly can have a mixed add/subtract/ignore status situation, in which case the status indicator in the Derived Assembly dialog is a three-segment, three-colour button to indicate the mixed status.
- In the Derived Assembly dialog box there is a Keep Seams… box. If it is checked, and if the assembly contains parts that touch each other as coincident planar faces, then the seams between them are retained in the derived part. If it is not checked (the default) then the parts merge into a single solid. You would normally only select this option if you want to use the seam as a reference for a later operation, but can't with the default setting because it has diapppeard.
- The Other tab of the Derived Assembly dialog box lets you choose exported parameters, unused sketches, work geometry, and surfaces to be carried forward into the new derived part.

As we have seen, derived parts are an extremely powerful tool and yet are remarkably simple to use.

Enjoy!

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