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Autodesk Inventor Tutorials

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Derived Parts
Part One - Weldments
Latest Revision: 5/06/02

One of the more popular questions I see on the Inventor Newsgroup is how to use Boolean operations in Inventor. While Inventor does not have direct access Boolean commands, **Derived Parts** is a feature that provides the same output as Boolean unions and subtractions.

Quite simply a derived part is a copy of another part or assembly. Furthermore there is a parametric link between these two parts. When the base part or assembly changes, the derived part is updated to reflect the changes. While there are countless uses for derived parts such as skeletal modeling, derived surfaces and part replication, the three most common uses for derived parts are mold creation, scaling and copying parts, and weldments. We shall cover the latter case first.

Simple weldments can often be created in one part (.ipt) file. However complex weldments are usually created in the assembly (.iam) context. (Creating weldments in assemblies also gives you the ability to detail the individual parts that comprise the weldment). After designing the individual parts that comprise the weldment you constrain the parts in an assembly. (see Figure 1)
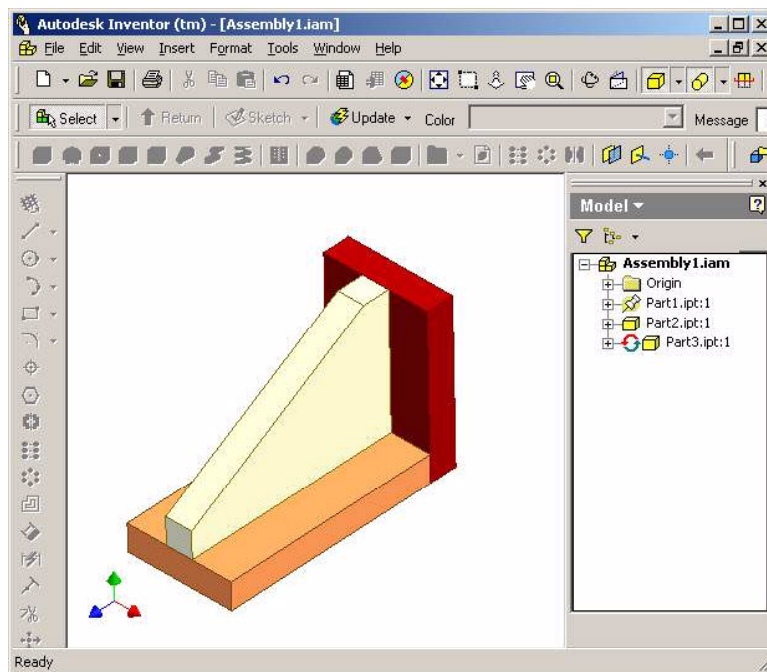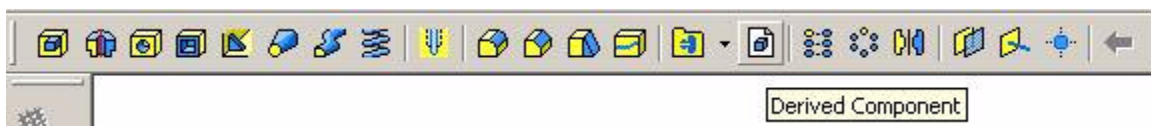

Figure 1 – Base Assembly

Next start a new part file. Exit sketch mode (if your template starts in a sketch) and select the **Derived Component** button in the **Features** toolbar.

You will be prompted to select a part or assembly. Select your weldment assembly. You are now presented with the **Derived Assembly** dialogue box. (See Figure 2)
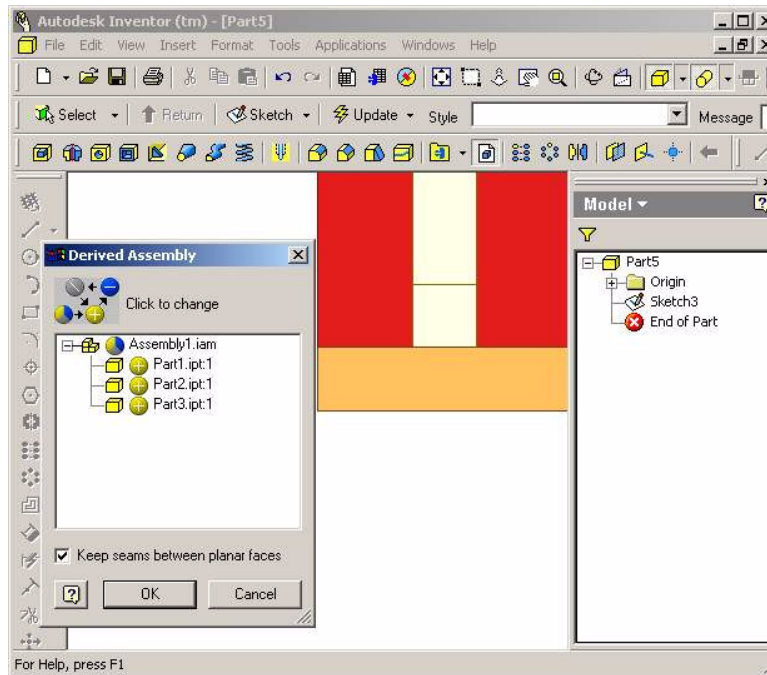

Figure 2 – Derived Assembly Dialogue Box

You must now choose which parts that comprise the assembly will be included in the derived part. This is where the Boolean operations come into play. By clicking on the yellow "+" signs you can cycle through three options

- Yellow Plus: Includes the selected geometry in the derived part (Boolean union)
- Blue Minus: Subtracts the selected geometry in the derived part (Boolean subtraction)
- Grey Slash: Do not Include the Part (but do not subtract)

The "Keep Seams Between Planar Faces" check box determines if lines will be draw between the faces of parts. Try toggling this option on and off to get a better understanding of the effect. (For weldments it is often better to check this box as it makes differentiation between parts more clear).

After exiting the dialogue the derived part is complete. (see Figure 3)

The derived part is a single solid body. Features can be added to this derived part but the base geometry cannot be changed within the context of the derived part. Any changes to the base geometry must be made from the base assembly. The browser tree shows the parts that comprise the derived part and their state (union, subtraction or ignored).

Let us now go back to the original assembly and make some modifications to the height of the red plate. Increase the height of the red plate. (see Figure 4) (In this example the tan triangular gusset is adaptive and grows with the height of the red plate. Adaptive

parts are not necessary to achieve this. To achieve this effect without adaptivity, simply modify the tan gusset as well as the red plate.)
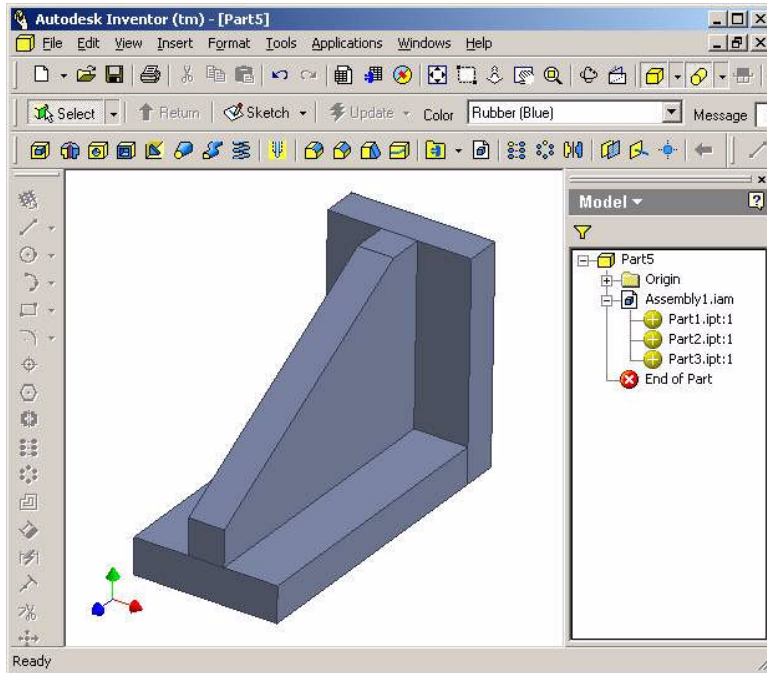

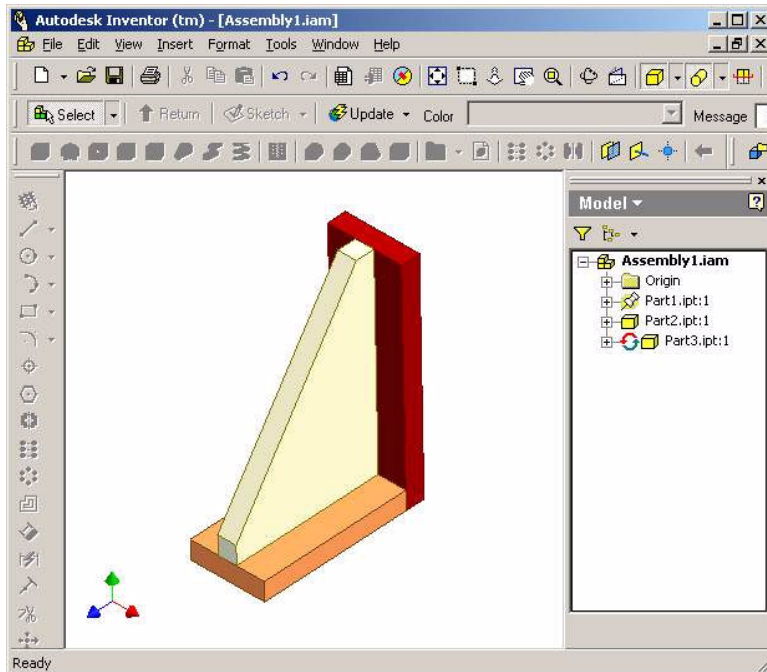**Figure 3 – Derived Part**

Figure 4 – Modified Base Assembly

After saving the updated assembly return to the derived part file. (see Figure 5). Notice that it has not changed in relation to our new assembly model. Look closely at the icon next to **Assembly1.iam** (or the name of your assembly file) in the browser tree. Notice that it has changed from the standard derived part icon to a red lightning bolt. This indicates that the base file has changed and that the derived part needs to be updated.
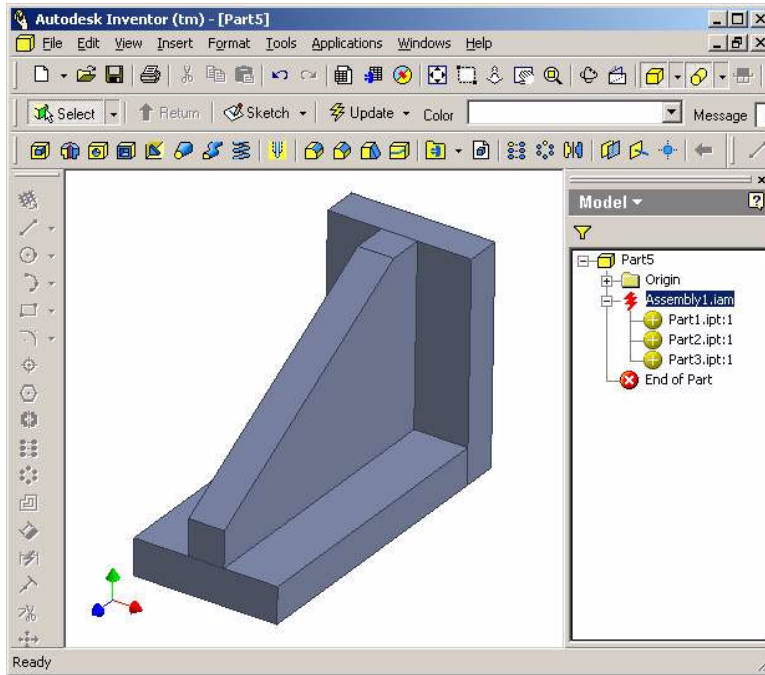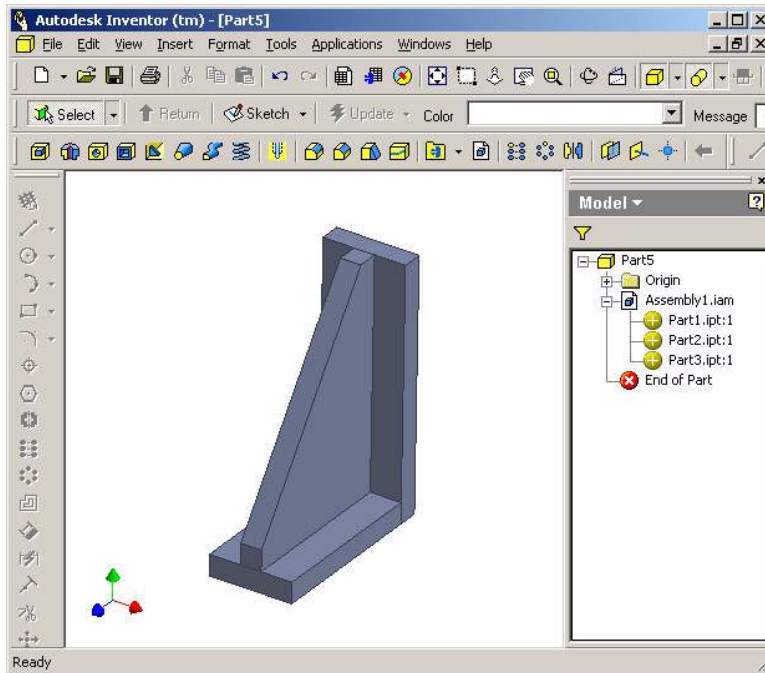

Figure 5 – Derived Part (before update)

Figure 6 – Derived Part (after update)

Click the Update button and the derived part will update the geometry as defined in the base assembly. Notice that the icon has changed back to the original derived part icon. The derived part is now "in sync" with the base file. (see Figure 6)

Now that the derived part is complete, additional features such as holes, fillets, and chamfers can be added to the derived part. This allows the user to have models that represent both pre and post machining operations.

In some cases it may be useful to sever the link between the derived part and the base file. This can be useful if there is the need to make several parts that are similar to a base assembly. For example, at the end of the lesson we have a "large" weldment. If a "small" weldment (as in the first step of our lesson) is desired, the user can choose to sever the link between this "large" weldment and the base assembly file. This "freezes" the state of the "large" weldment derived part. After changes have been made to the base assembly model a new derived part can be made based on the "small" weldment assembly file.

To sever the link right click on **Assembly1.iam** (or the name of your assembly file) in the derived part browser. From the context menu we choose *Break Link with Base Assembly.* This severs the parametric link with the base file. Any changes made to the base assembly will no longer be reflected in this derived part file. (***NOTE: Once severed, a link cannot be reestablished with the base file).***

This procedure provides a fast way to make several similar parts from a base assembly.

In the next part of this article we will discuss other Boolean operations and how they pertain to mold design.