

## Autodesk's Inventor Fusion takes a different path to direct modeling

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Recently we had the opportunity to spend some time with Jay Tedeschi, Autodesk Product Solution Evangelist. Jay visited us at TechniCom headquarters in NJ and we spent a couple of hours exploring Fusion and its integration into Inventor.

### It works best like this

You can begin to model either in Inventor Professional or Fusion. But it is most interesting to consider its place in the workflow. An ideal solution is to start within Inventor, switch to Fusion for some direct modeling, then return to Inventor to finish the model.

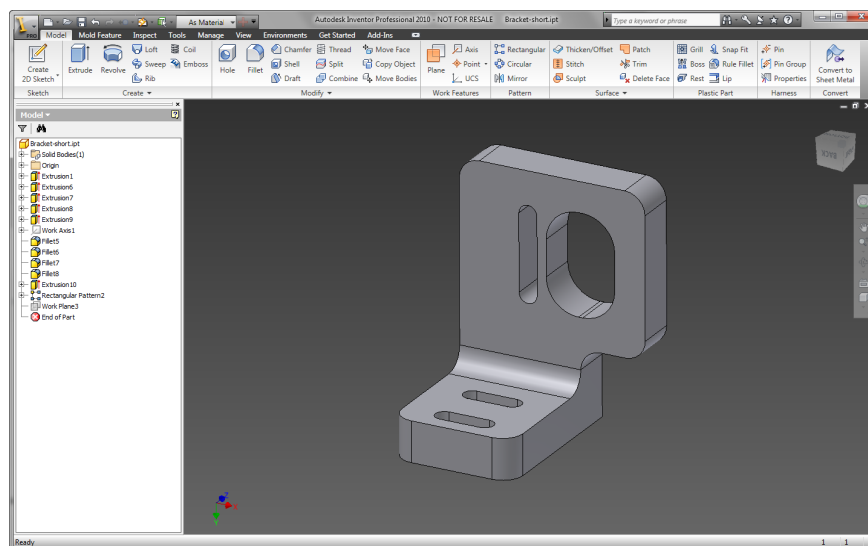
The key is what happens next. All other modeling systems choose to leave the direct model alone, perhaps treating it as a b-rep solid in which its build history (or recipe) is lost. However, Inventor tries to convert the resulting direct model into a parametric (history based) model. We wanted to see how successful it is at doing so.

Beginning with a part in Inventor, you close that part in Inventor and open it within Fusion. Now you can perform direct modeling of the part within Fusion. Fusion builds a list of the operations performed, but they are not in any specific order and thus not parametric. Clicking on an operation allows one to edit the operation itself.

When finished with Fusion you save the model as a .dwg file, then open that file within Inventor using the Change Manager. Visually, one sees the original Inventor part and the changes made to that part highlighted in a yellow translucent form overlaying the changes to the Inventor model.

We went through quite a few models investigating what kind of changes could be made in Fusion, how they could be brought back into Inventor, and how it might effect the original Inventor part.

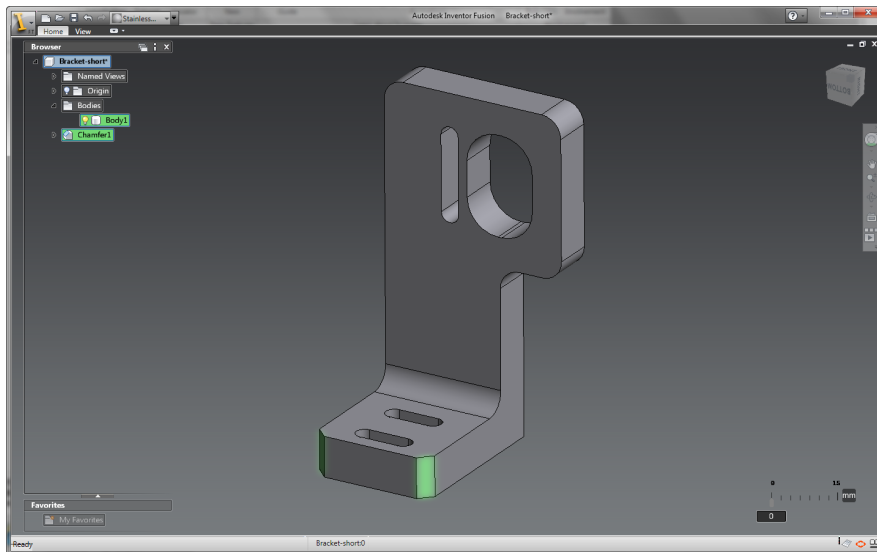
In one case, we worked with a right angle bracket with some slots and fillets originally created in Inventor.



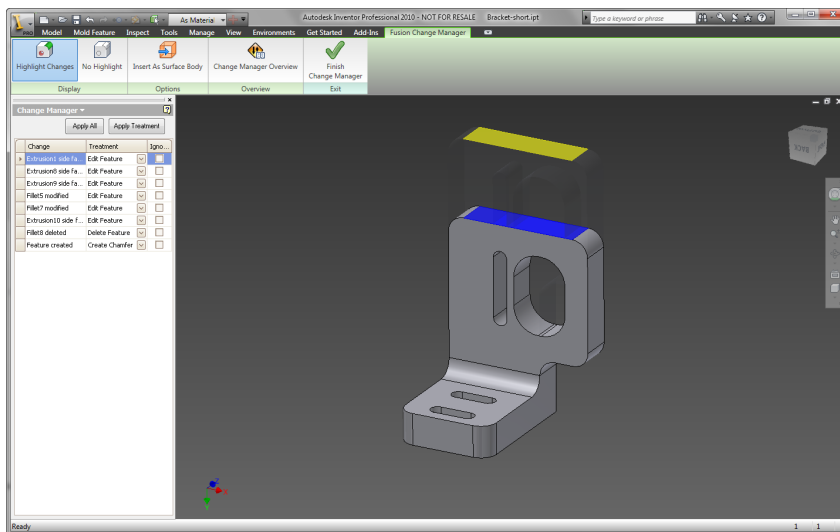
We read the part into Fusion and selected the face with the slot. We dragged the face to make it bigger and Fusion kept the slots in the same position relative to the

top edge. Then we deleted two of the lower fillets. It was interesting – deleting the fillets caused the former faces of the fillet to revert to sharp edges. How we wondered, especially if the original fillet feature was not present? The explanation was that Fusion tries to keep a closed volume. Fusion builds a representation of the operations performed, but they are not in any specific order, and are thus not parametric or history based. You are able to click on them and perform various functions within the operation or edit the operation itself.

Here is an example of that below. Note that the tree on the left depicts the operations, but are unordered. This is kind of handy because, as in the case shown, selecting the chamfer displays the chamfer on the part and allows editing that operation, perhaps to change the size or angle of the chamfer.

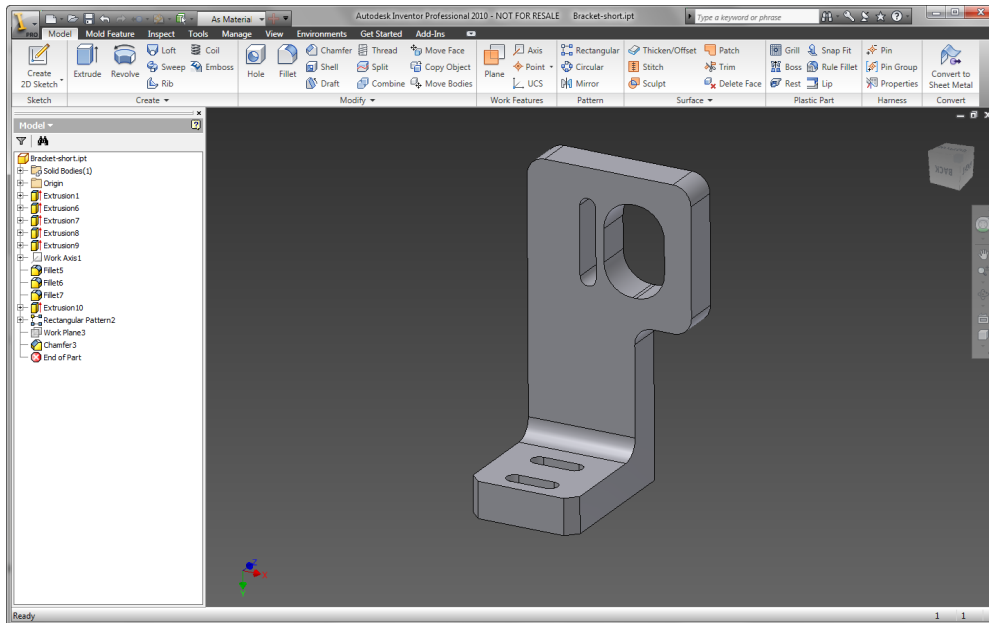


We changed the now sharp edges to chamfer by using the glyph and then changed the size of the chamfer edges. We then closed the Fusion model, saving it with a .dwg extension. Then, within Inventor, we open the fusion part. Inventor now displays the original Inventor part with the changes made to the model shown in translucent yellow, as shown below. It is a bit difficult to see, but the relocated slots and the chamfers are shown in dimmed yellow in the image below.



## Viewing the .dwg part within Inventor

The change manager menu on the left allows applying all the changes or accepting individual changes. Selecting all of them results in this change to the model, which looks exactly like the changes we made in Fusion. **Not only that, but the history tree contains a history of the changes made to the model in Fusion. Note the addition of the chamfer to the history tree and the changes made to extrusion 10.** What happened was that the Change Manager changes the Inventor history to match the changes made within Fusion.



Voila! Here it is in Inventor. It seems like magic!

We also had a chance to speak with Kevin Schneider, the Autodesk architect of Fusion and Change Manager. Here are his responses to a few interesting questions. [Kevin was given an opportunity to edit this Q&A dialog but was not able to respond in time for this post.]

### An interview with Kevin Schneider

Q. Where are you going with Inventor and Fusion in the future?

A. Both Inventor and Fusion are evolving from what you see today in the 2010 version. As both merge and they become one, the parametric system will evolve to make this transition easier. It will become a seamless, bi-directional model transfer from history-based systems to direct modeling and back. Changes made in Fusion will be converted, where possible, to parametric features. Where not possible to convert to parametric features, those changes will use a sculpt feature to generate a non parametric solid from the Fusion model's faces, thus allowing any change made in Fusion to be brought back into Inventor. In fact, this works today.

Q. Where does this stand today?

A. Today the change manager does not handle all features. We are continuing to work on the change manager technology to make sure it handles all possible changes. Features handled today include: changes to and new extrudes, fillets, chamfers, patterns, and hole features. In the future we will allow loft edits, sweep edits, revolve, and more complex edits to patterns. Our intent was to get Fusion out

early using Inventor Labs to get customer feedback. We are not yet finished with the technology to read the whole plethora of changes back into Inventor.

Q. Could I read a Fusion model back in today, somehow, even if not directly using the most efficient features?

A. When you try to bring these Fusion changes back into Inventor there are certain Inventor parametric features that we have not completed the round trip with yet. In these cases, you can always extract faces within a Fusion model and create a solid within Inventor using the SCULPT feature.

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