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Summary: Autodesk releases a second preview of Inventor Fusion Technology, and now we are convinced: the “fusion” is of AutoCAD and Inventor. Randall S. Newton updates the popular “Dark Nights” VEKTORRUM column from February. (October 29, 2009)

VEKTORRUM:

Dark Nights Of Intrigue at Castle Autodesk Chapter Two, In Which An Old Herb Becomes The Key Ingredient to Fusion Potion

By Randall S. Newton
Editor-in-Chief

Editor's Note: This VEKTORRUM column is a follow-up to "[King Carl, Prince Amar, and the Fat Lady: Dark Nights of Intrigue at Castle Autodesk](#)" published February 5, 2009. We will spare you the laborious Dark Ages scenario of the first installment.

October 29, 2009—I'm on vacation this week, so this will be short. Having said that, I am reminded of what the great mathematician Blaise Pascal said about writing. He loved to write letters to friends; once he wrote (this is a paraphrase), “I am sorry this letter is so long. If I had more time I would have written a shorter letter.”

In energy research ‘fusion’ is about more than pushing two atoms together, it is about finding a fantastic resource that delivers like nothing before it. In the CAD world we have ‘Fusion,’ the new design technology from Autodesk that is currently in technology preview at Autodesk Labs. The goal is similar; to develop a fantastic resource that delivers like nothing before it ... at least in the eyes of Autodesk.

When we first reported on Inventor Fusion technology, we speculated that the future of Inventor could be found in AutoCAD. Now we are certain.



During the demonstration of the new stuff in Inventor Fusion, I noticed that the demo jock was using the same model over and over, an AutoCAD DWG file. I also noticed that some of the new User Interface features look remarkably similar to new tools in the latest release of AutoCAD. At the end of the demo, I asked why. The answer from product manager Kevin Schneider was straightforward. DWG, he said, is the most consumable file format in CAD, it offers unique interoperability (a statement we won't question in this article, although it deserves to be challenged). This new release of Inventor Fusion also supports CAD files from Pro/ENGINEER and CATIA as well as the neutral STEP format, and that the use of DWG is the key.

At that moment it became obvious to me. Inventor Fusion should be more accurately named AutoCAD/Inventor Fusion. Thus the headline of this article. The old herb is AutoCAD, and it is the not-so-secret ingredient that makes Fusion technology possible.

I thought I should double-check my assumptions, so I used Facebook email (which always seems to get noticed) and asked Sean Dotson what he thought. Dotson is a manufacturing engineer and proprietor of the popular SDotson.com Inventor user support site. His view:

It's all merging. I think everyone has speculated on this but I see the products becoming one in the near future. Look at the icons and some of the commands. How AutoCAD now has geometric constraints. Grips in Inventor is akin to what boolean solids in AutoCAD used to be.

There is some good and some bad to this. AutoCAD sketching is superior but it's all bogged down with a lot of baggage. Inventor would bring true 3D to AutoCAD. DWG will become the universal file format for 2D and 3D. And things will get way more complicated.

My guess is they will be one program in less than three years.

What's in the Latest Version

Autodesk says this second Inventor Fusion Technology Preview offers new features to "unite direct and parametric modeling workflows within a single digital model created in Autodesk Inventor software."

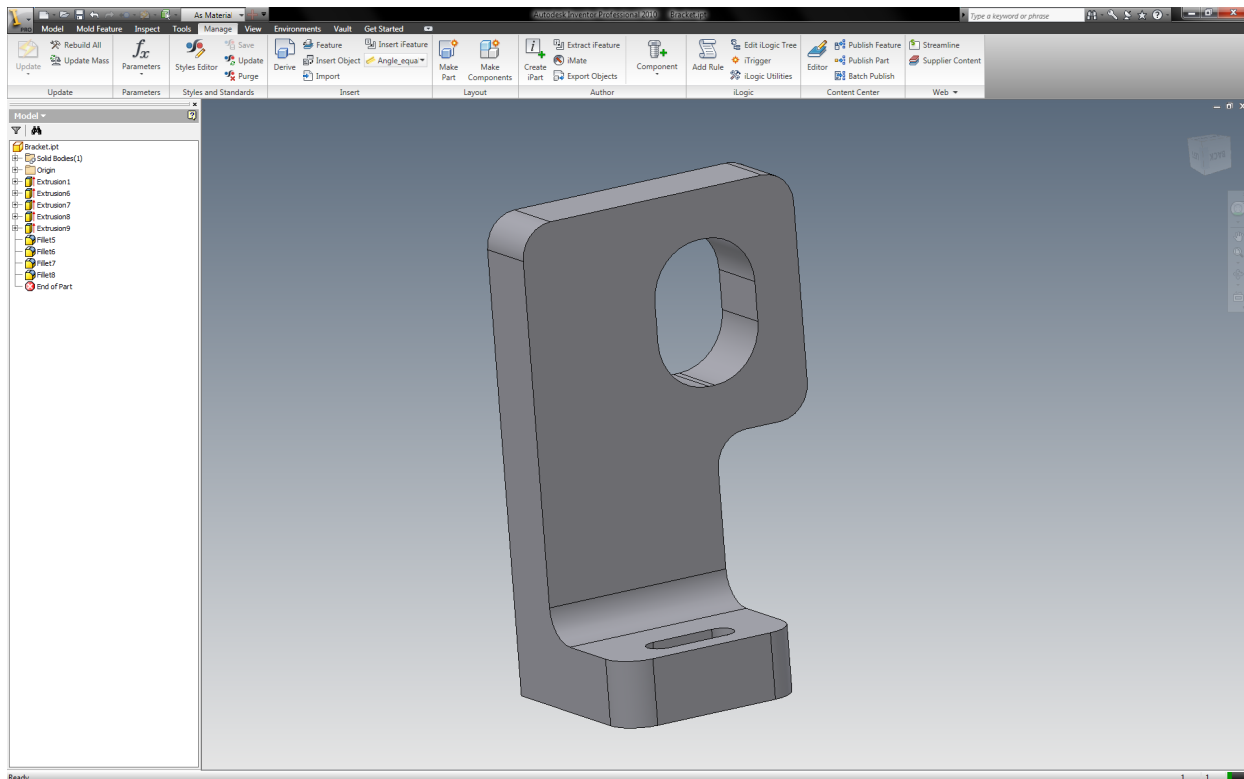
Specifics include:

- Change Manager functionality, which enables users to edit a model in Inventor Fusion and then move it into Autodesk Inventor software, automatically updating the model if the user decides to accept the changes. Readers who plan to test this should note: Inventor Fusion change manager functionality is currently only included with the Inventor Fusion

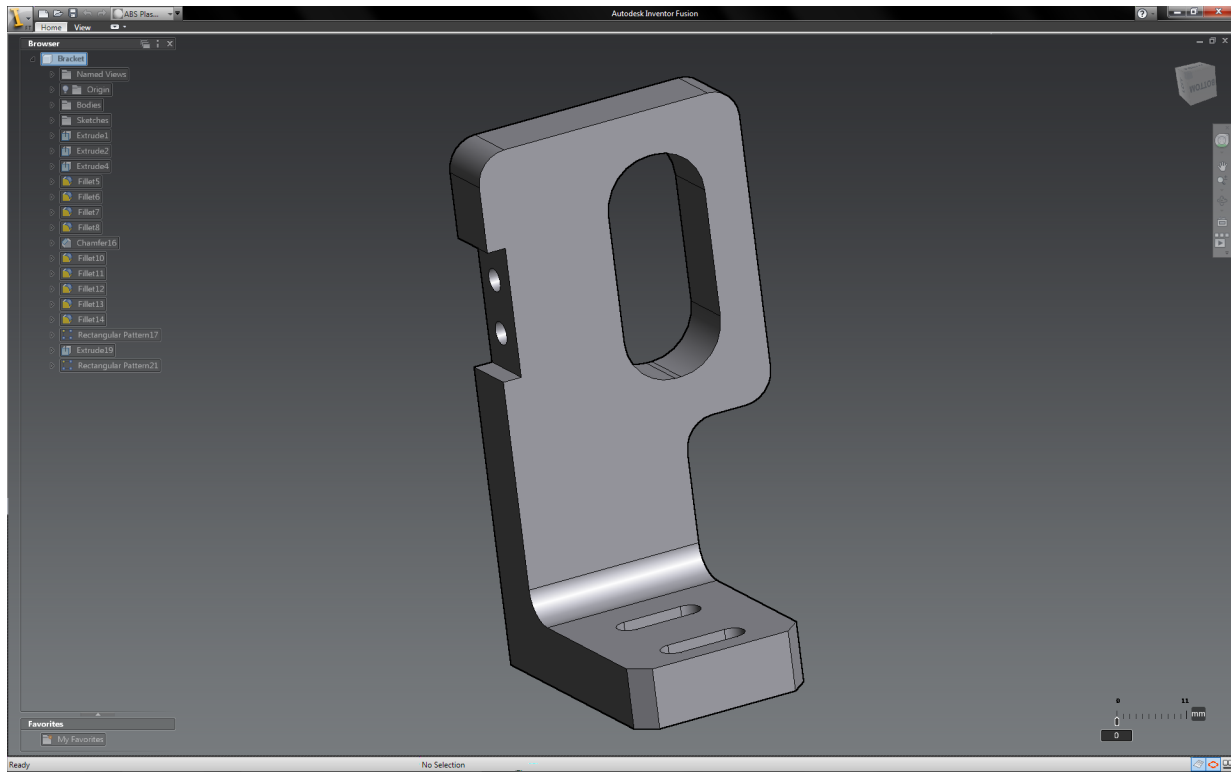
Technology Preview download. It applies only to original Inventor data and can be activated only if Autodesk Inventor 2010 (Subscription version) is installed.

- A new moderated [Wiki Help site](#) that enables users to more effectively collaborate, troubleshoot and share ideas and reference materials. Users can add or edit help information and share videos, best practices, tips, tricks, tutorials, comments, tags and Web widgets.
- Refinements and enhancements to the user interface, sketching and modeling tools, and underlying graphics technology.

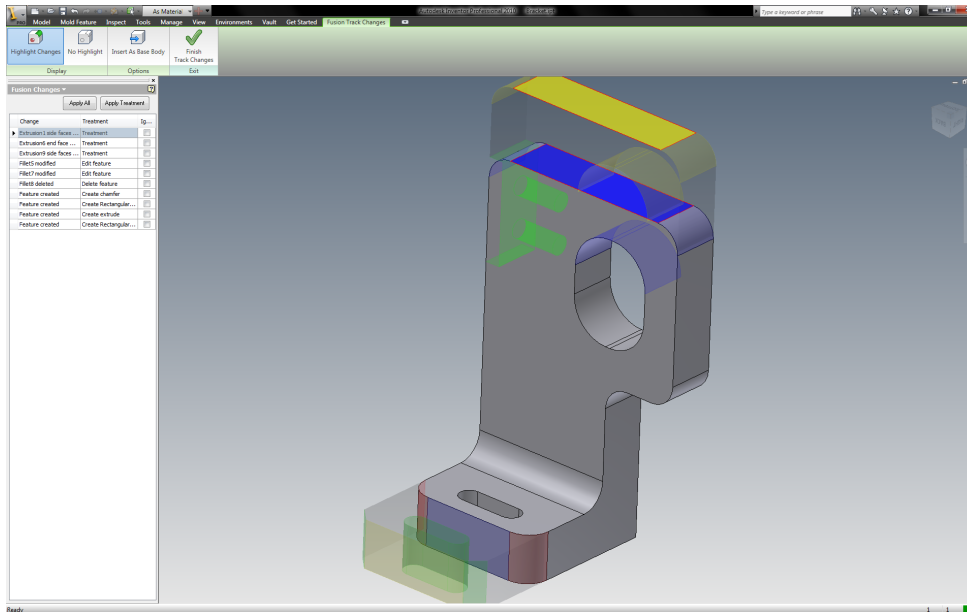
The Change Manager functionality is straightforward. The user makes changes to an existing parametric model in Fusion, using direct editing. The directly-modified model is then opened in Autodesk Inventor. The Change Manager color-code highlights the changes, and allows the user to selectively accept or reject changes for import into the Inventor's history tree.



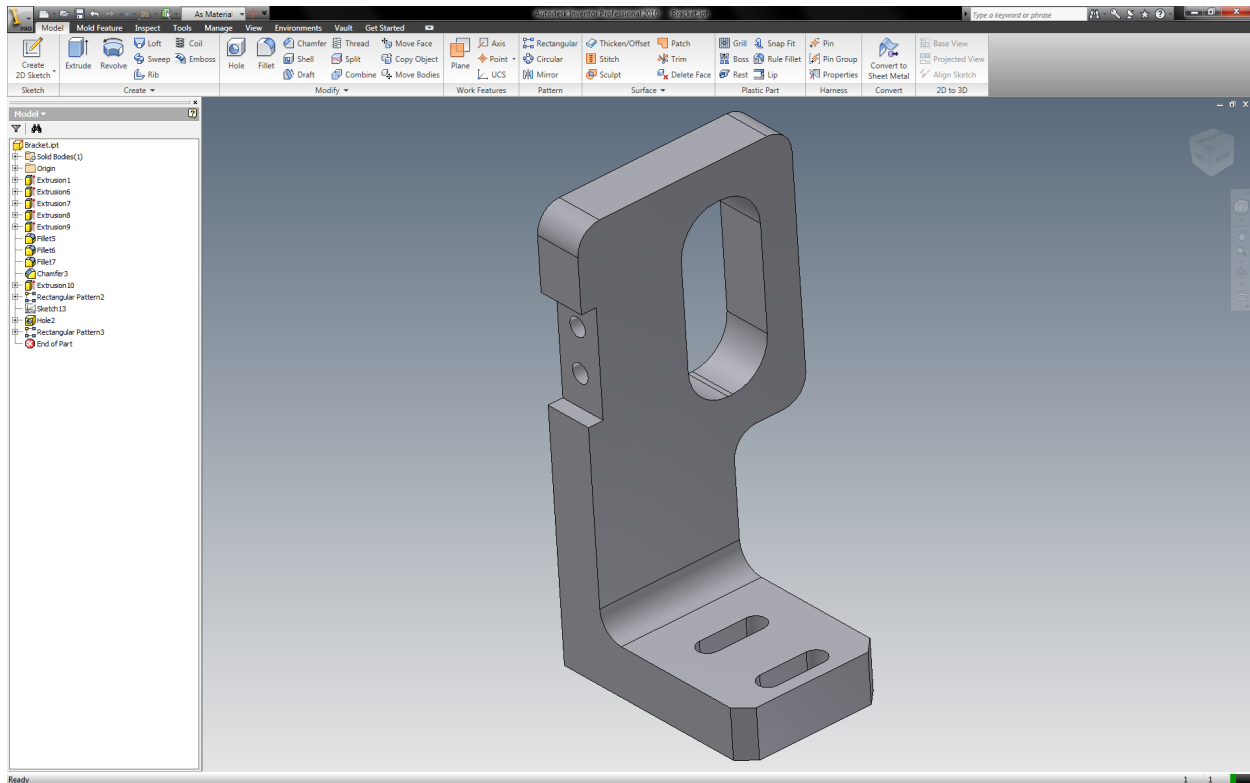
Step 1: An existing Inventor model is opened in Inventor Fusion, as a DWG.



Step 2: Using direct editing, (as if you were doing it in AutoCAD) changes are made to the model.



Step 3: The revised model is opened in Autodesk Inventor; changes are highlighted.



Step 4: Changes are accepted and written into the history tree at the spot where the original geometry is listed—not appended to the end of the list.

The Fine-Print Details

The Inventor Fusion Technology Preview is available at labs.autodesk.com. The preview download is currently available in the following countries: Australia, Austria, Belgium, Brazil, Canada, China, France, Germany, Ireland, Italy, Japan, Luxembourg, New Zealand, Singapore, South Africa, Switzerland, United Kingdom and United States. Autodesk may also expand availability to other locations in the future.

For questions or comments please contact info@cadcamnet.com.

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