



Tip and Trick Week of 06/20/10

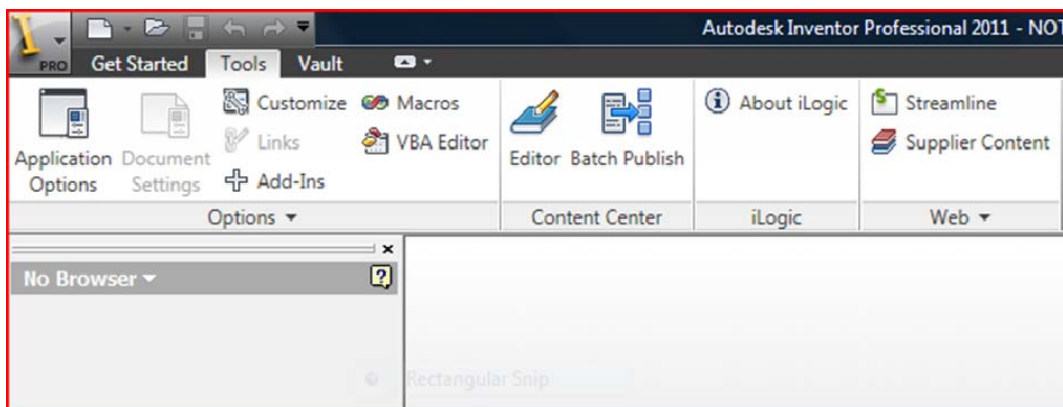
How to Automatically Leverage Inventor Fusion through Autodesk Inventor 2011

One of main functions of Autodesk Inventor Fusion (Preview 3) is to allow users to work with and manipulate other CAD files/neutral file formats such as: .stp, .sat, .prt etc. Having this ability goes a long way for a designer to communicate data inside and outside the walls of their facility. Within Autodesk Inventor 2011 there is a direct connection to Inventor Fusion Preview 3 and this tutorial will show you the steps of how to set that up.

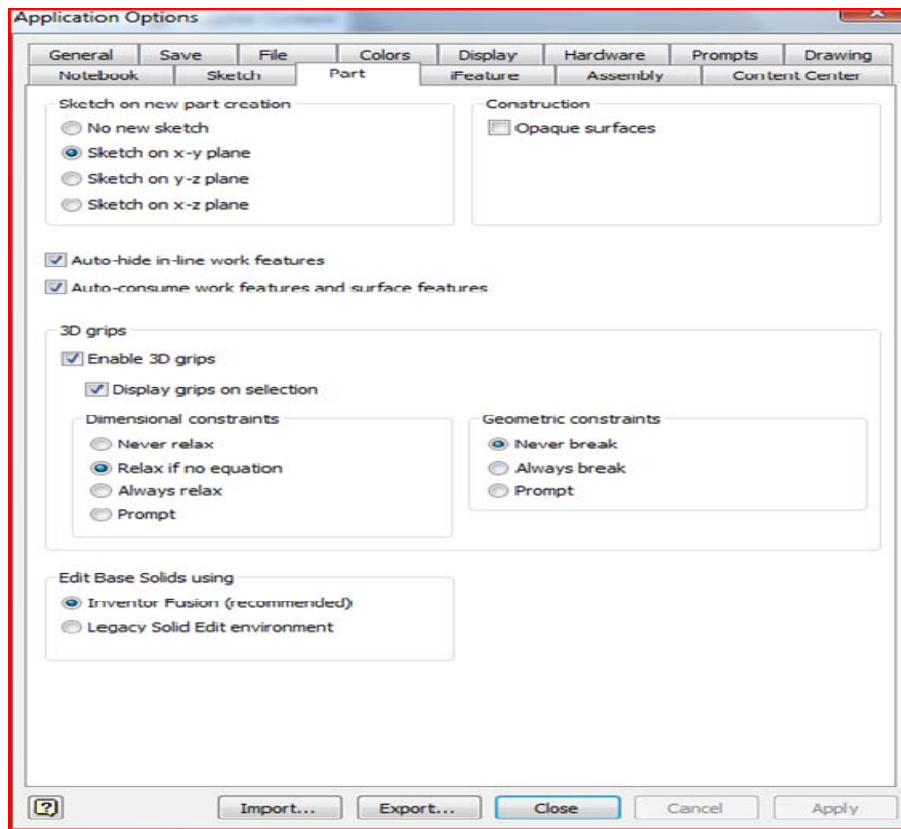
The first step is to make sure Inventor Fusion Preview 3 is installed on your machine. This can be done by downloading it from <http://labs.autodesk.com/technologies/fusion> or if you have the Autodesk Inventor 2011 media, there is a DVD for the installation of Inventor Fusion.

Within previous releases of Autodesk Inventor, if a user wanted to attempt to import and clean up a neutral file format they used the construction environment to analyze/clean up the imported file. Still, when trying to make any edits to a solid base this is still the default setting with Autodesk Inventor 2011. This needs to be changed so users can leverage the power of Autodesk Inventor Fusion.

Within Autodesk Inventor go to the Tools Tab>Options Panel>Application Options. See image below



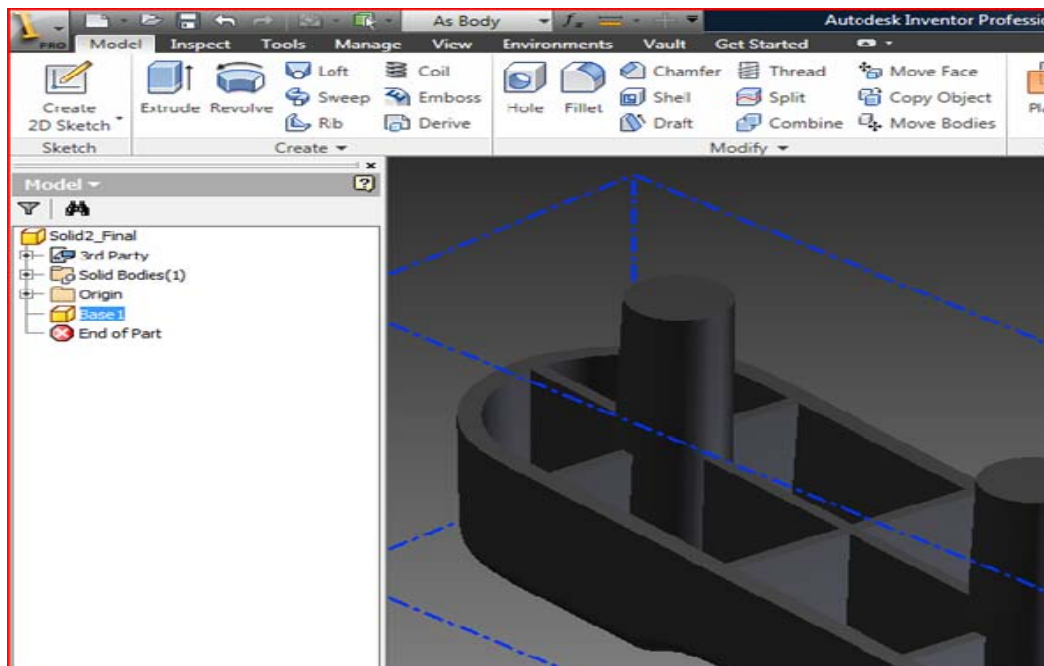
Once in the Applications Options dialog box, select the Part Tab. At the bottom of the Part Tab there is an option for Edit Base Solids using, there are two options, select Inventor Fusion (Recommended). See image below



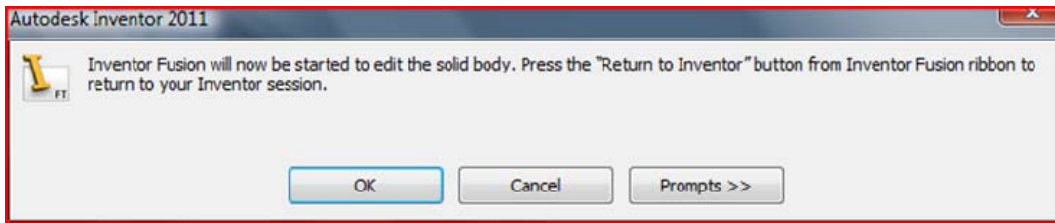
Notice Legacy Solid Edit environment is selected; this is the default when Inventor 2011 is installed.

Now, instead of activating the construction environment within Autodesk Inventor when editing a solid base, this will automatically launch Inventor Fusion and load the file into the session for editing.

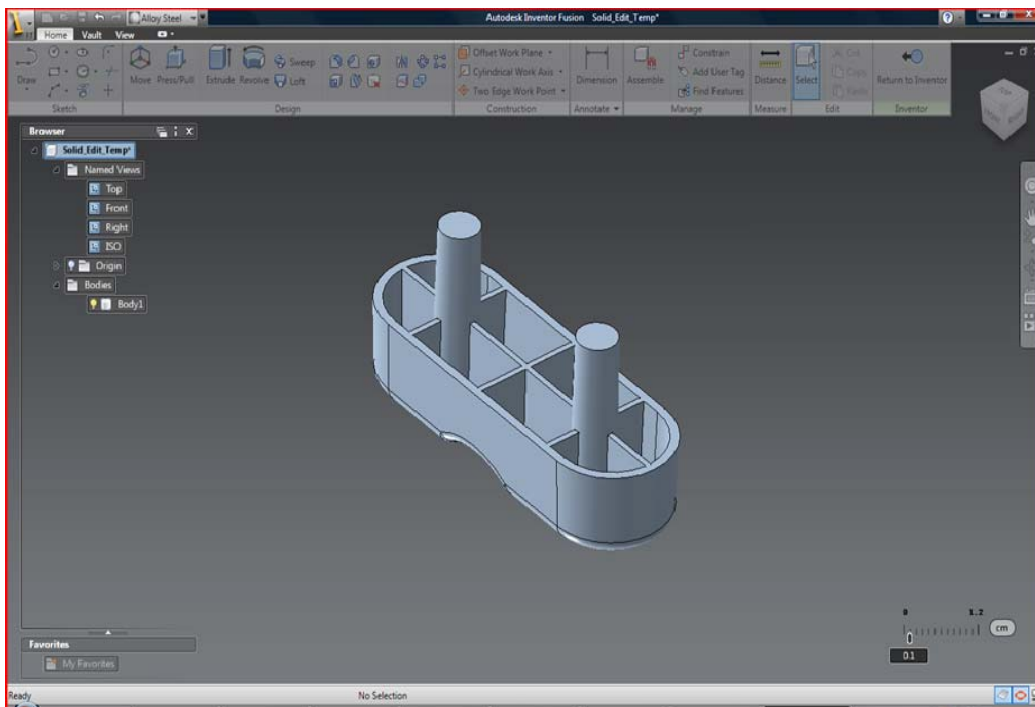
Right click on the Solid Base within the Autodesk Inventor browser and select edit base, this will launch Inventor Fusion. See image below



The following message will display alerting you that Inventor Fusion is going to launch for the editing. See image below



Select the OK button to launch Inventor Fusion. This will launch Inventor Fusion and load the file. Note: When Fusion launches, you may need to select Zoom Extents to make sure the model appears in the Graphics Window. See image below.



Now the changes can be applied. Any changes that are made within Inventor Fusion will correspond back to Autodesk Inventor. When you want to return to Autodesk Inventor to continue working, select the Return to Inventor button on the Ribbon Bar. See image Above