Autodesk Products Tip & Tricks

Product: Inventor 2008
Topic: Redefining the Isometric View in Inventor 2008
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Since I started working with Inventor, one of the questions I tend to get from customers is how to re-orient their isometric views in the 3-D environment of an .ipt (part) or .iam (assembly) file.

The isometric view is accessed in several ways:
1. Right Clicking in Space and selecting Isometric View
2. From the Standard Toolbar, View→Isometric View
3. F6 on your keyboard

If I orient the assembly into its isometric view, I see the result shown in Figure 1. This may not be the desired orientation.
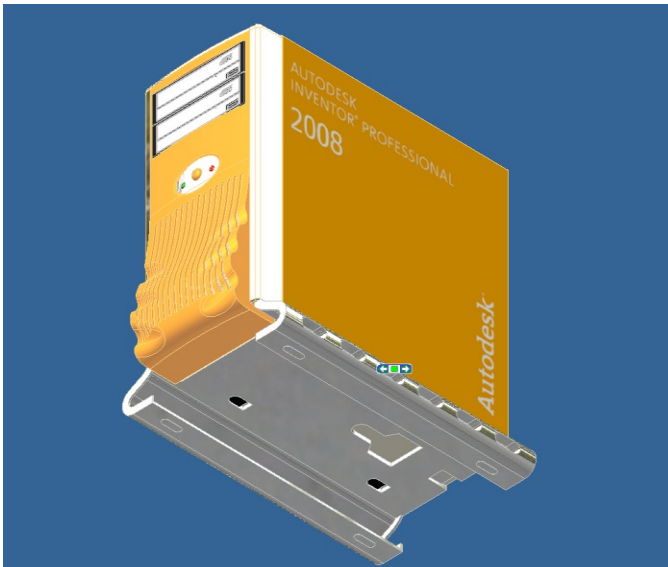

Figure 1

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To change this default isometric view into a more visually representative illustration, simply follow these quick and easy steps:

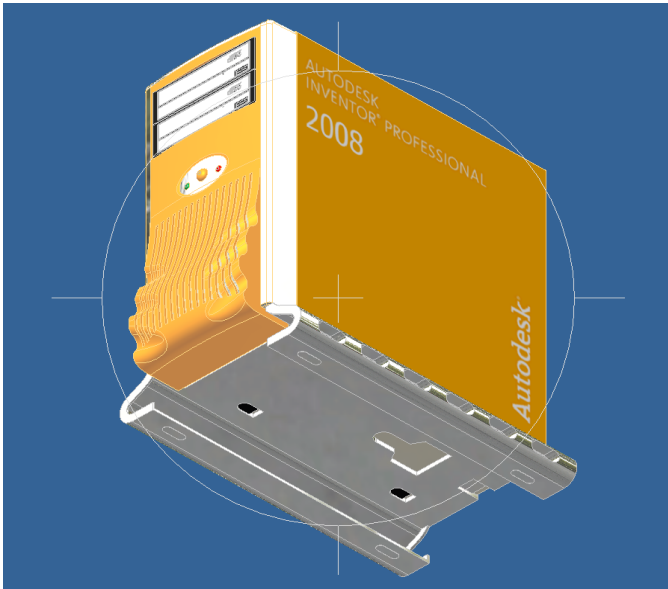➢ Start the Rotate tool (Figure 2).

Figure 2

➢ Switch from your Free Rotate View to your Common View, or what is sometimes referred to as the 'Glass Box' View, by Right-Clicking→Common View or simply hitting the space bar (Figure 3).
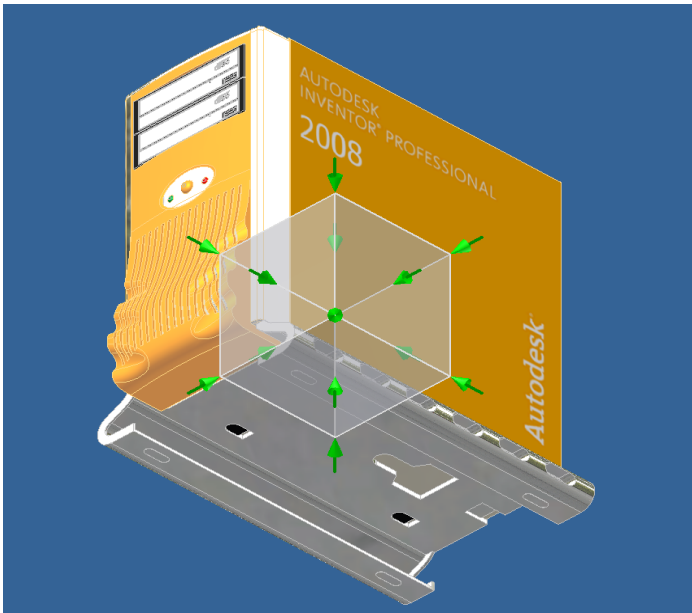➢

Figure 3

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> ➢ Set your desired Isometric View by selecting the corresponding green arrow. Once you happy with the new default isometric orientation, Right Click on green arrow and select 'Redefine Isometric' (Figure 4).
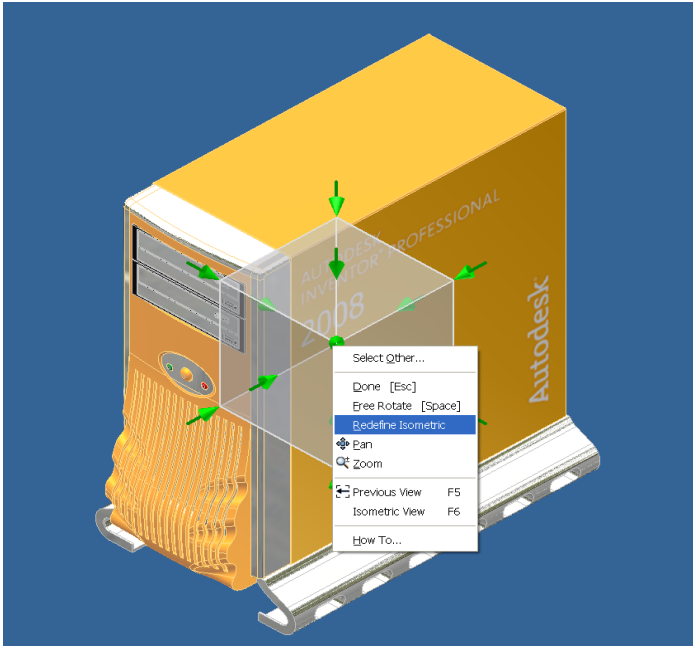

Figure 4

From now on, whenever the isometric view is accessed, it will position itself into the new orientation we just defined.

Thank you for visiting our website and accessing our tips and tricks. I hope this will benefit you in the future and give you something to brag about to your fellow co-workers.