Autodesk Inventor Tutorials

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Tips for Assemblies

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I was recently helping a small group of new users learn how to effectively use assemblies. As I went thought my presentation I realized that I had given this same information to many users in the past. I decided it was time to get it all down on paper. Kind of a "Assemblies for Dummies" if you will ☺

Here is a collection of general assembly techniques. Some may help you out, some may not. You may even disagree with some of them. Just take them all with a grain of salt. If you apply them in some fashion or another however, I do feel that it will make your time dealing with assemblies a bit easier.

1. Build Your Assembly

The biggest piece of advice I can give to users is to build your assemblies as if you are actually building your designs: **Constraint things off logical positions**. If you are placing shaft into position, constraint it to the bearings. If you are placing a cover over an opening, use the surface of the opening to mate your cover into place. If you are bolting two plates together use an insert constraints on the boltholes to align these plates.

I see many users constraining off far sides of a machine or off random workplanes just because they are convenient. While it make take a second or two more to do it correctly, in the long run (and when things go wrong) it makes it much easier to figure out what constraints are in conflict if you build you assembly in a logical manner.

Begin your assembly with the logical base of your machine/item/gizmo. In machine building this is often a base or tabletop. When you being you design with a bearing or some random plate, all constraints will grow from these parts. It's better to have a tree-like constraint structure (many constrained to one) than a straight line (parts constrained together in a chain)

2. Use Sub-Assemblies

Following the theme of #1, break your design down into smaller pieces. I have seen assemblies with 5000 parts. There is often no need for this. In this many parts there must be an opportunity to group parts into subassemblies.

The use of subassemblies not only allows you to turn off sections of the design that are not in use but also breaks down groups of constraints into manageable chunks. Once a sub assembly is fully constrained it only takes 6 degrees of freedom (often no more than 3 constraints) to lock it into position in the top-level assembly.

Furthermore, subassemblies allow you to reuse design data multiple times. Instead of patterning parts in an assembly (which can be difficult to change), demote the parts into a subassembly and insert the sub multiple times.

3. Be Consistent

When inserting the same part multiple times, be consistent. Let's look at Figure 1. In this design there are three instances of the rack assembly. Each rack is fastened to a plate that has four countersunk screw holes (see inset). To constrain these assemblies to the blue frame in the top level assembly I chose to always use an insert constraint on the top left and bottom right holes. I did this for all three sub-assemblies. In fact, in general I do this for all plates with perimeter mounting holes. By being consistent if anything went wrong with one of the plates I would know how it was constrained by looking at the other two. By placing consistent constraints you no longer have to remember exactly how you constrained an objects as long as you know how you "usually" constrain such an item.

Another example of this technique is the order in which you pick your constraints. For example, some users always pick the bolt head as item #1 and the hole as item#2. This type of consistency will help you when things do (and they will) go wrong.

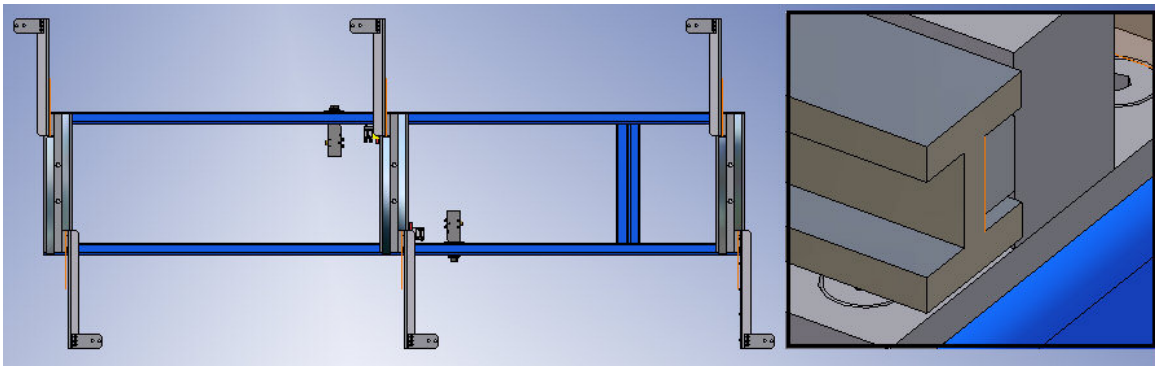

Figure 1 - Consistent Constraints

4. Cheat a Little

Often when you are trying to select a face (especially a skinny one) it seems like no matter what you can select everything **but** that face. A trick I learned back in R1 is to change your constraint type to flush (which only selects faces, not edges or points) and then select your faces. Now switch back to mate and apply your constraint.

Be aware that if you have preview turned on, your part may flip around a lot.

5. Use Adaptivity in Logical Places (Then Turn it Off)

Too often I see new users who have discovered Adaptivity go off the deep end. Adaptivity is a very powerful tool but it can be used incorrectly (and excessively). If you have a part with holes in it (part A), constrain the mating plate (B) to this part in

position. You can then project the geometry of the holes in part A to part B. Depending on your adaptivity settings these holes will now update to follow the position of the holes in part A. Very handy…but taxing to your model if you have 100s of adaptive holes.. If you know that the position of the holes in part A are not going to move, why should the holes in B be adaptive? Turn off the adaptivity. You can always turn it back on at a later point in the design.

Another example is using adaptivity to size a plate. You can sketch a general shape and the insert the part into the assembly. Make it adaptive then by using constraints, size it to the desired shape. This allows you to design without having to make constant measurements. However once the plate is sized and the design is semi-concrete, turn off the adaptivity. If the geometry on which the part is based ever changes, turn adaptivity back on and allow the plate to resize.

Finally, adaptive sub assemblies, while necessary to show sub assembly motion, can be VERY taxing on your system. I suggest turn off adaptivity on these subs until you need to move them. Once you have moved them into the correct positions or have checked the movement, turn off adaptivity.

6. Whenever Possible Use Insert

The insert constraint is a very powerful tool in that it's two constraints in one. It's both a face-to-face mate and an axial mate built into one constraint. It removes 5 degrees of freedom with one constraint. In Figure 2 you can see that we can fix the position of the cover with two Inserts on the corner holes (A-A and B-B) as opposed to 3 mate/flush constraints. In addition many times you do not care about rotational degrees of freedom (e.g. you aren't going to fix the angle of a bolt) therefore one insert constraint is all that is needed.

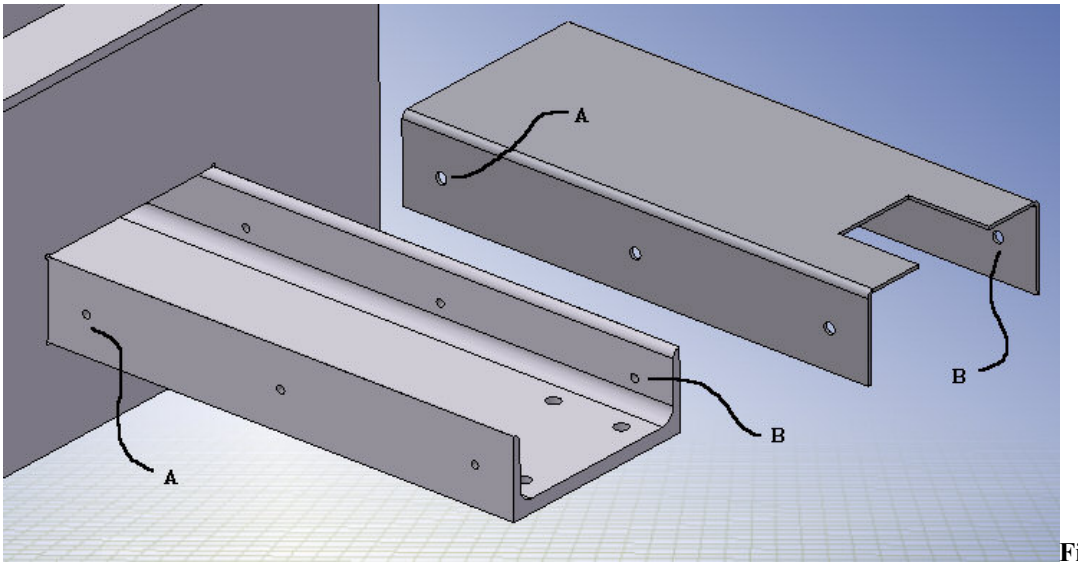

Figure 2 - Insert Constraints

7. Daisy-Chain Constraints

While it may seem to go against tip #1, it is often a smart move to daisy chain you constraints from part to part. A good example might be a fence (we'll ignore the fact that you could do this with a pattern to make our point). Instead of mating each picket to the fence stringer (see Figure 3), mate the first picket to the stringer then use a flush on the front side of the pickets to mate each successive picket to the first. This way if you ever need to radically change the stringer's shape (or replace it) you will only loose one constraint instead of 10s or 100s. The same technique can be used to position the pickets in the vertical direction. Constrain the first picket to the stringer and then make the bottom of each of the other pickets flush with the first.

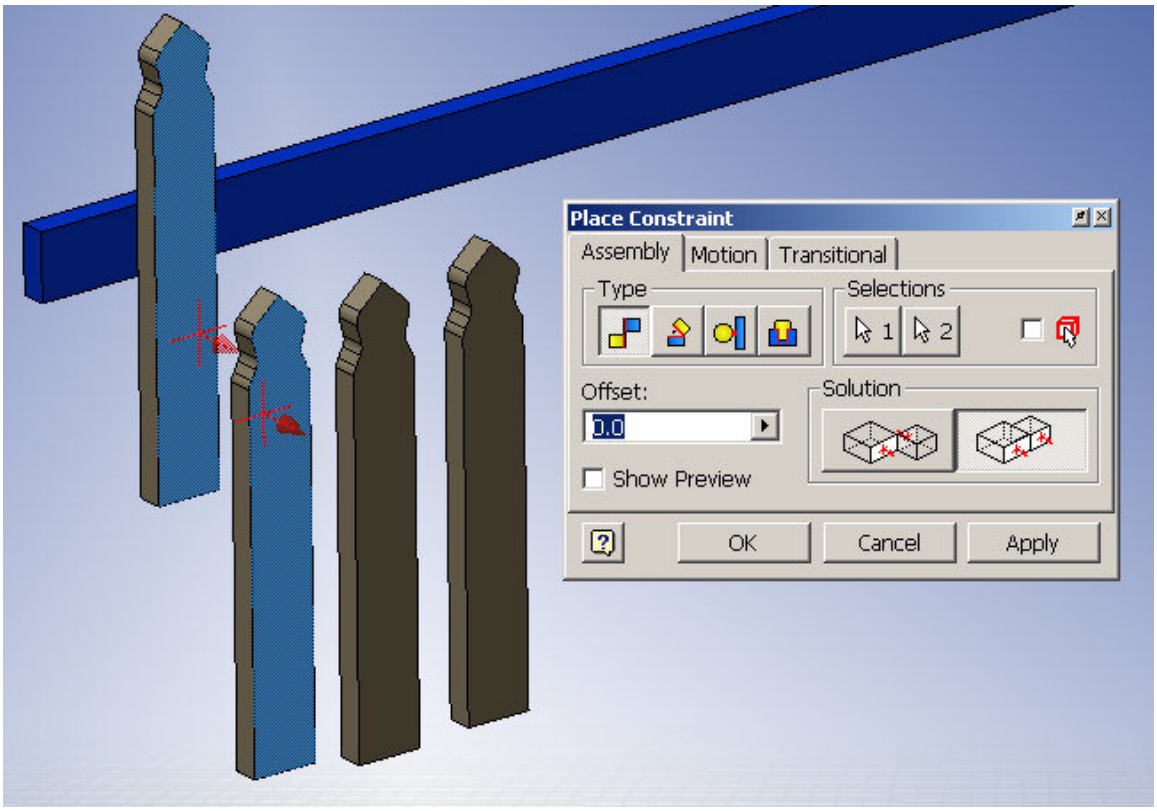

Figure 3 - Using Daisy-Chained Constraints

8. Don't Loose Those Constraints

Let's say you have a part that is the very foundation of your assembly. Everything is constrained to it. Then you realize that this part will have to be different from the standard version and you'll need to replace it. Not to worry!

Open the part to be replaced and "Save Copy As.." to the new name. Close the original file (without making any changes to it) and return to the assembly. Issue the Replace Component command on the original part and select the newly saved copy as

the replacement. It will look identical to the old part. All of your constraints will remain intact. This only works however if you **do not open the copied part before replacing and make no changes to the original after you have saved off the copy**. This works because you are fooling IV into thinking this is the same part as the original and the part is not dirtied.

9. Don't Panic When Things Go Wrong

One of the hardest things to explain to a new user is how to recover from a massive explosion of constraint failures. One thing to remember is that Inventor evaluates every constraint each time one is applied. This means if your model is fine and then you place a constraint, which causes 10 failures notices, that last constraint was the culprit. While the constraint may be valid by itself, it may be causing something else to move that causes the problems. So you can therefore cancel out of this one constraint and search to determine what other constraints are being affected by this one.

Another technique is to accept the constraint warning. Next switch to Modeling view. Do this by selecting the pull down (see Figure 4) and selecting Modeling view from the list. This places all constrains into one folder at the top of the browser.

Next find the constraints with errors (with a yellow "!") and suppress them all. Now turn them back on one at a time and see which one causes the rest to fail. Once you find the one that relives the errors you can delete or modify it. Sometimes you may have to repair more than one constraint.

Another helpful tip is to hover over the icon for the constraint (in either Position or Modeling mode). You will be presented with a tool tip like in Figure 4. This lists the type of constraint and the display name of the other part affected by the constraint.

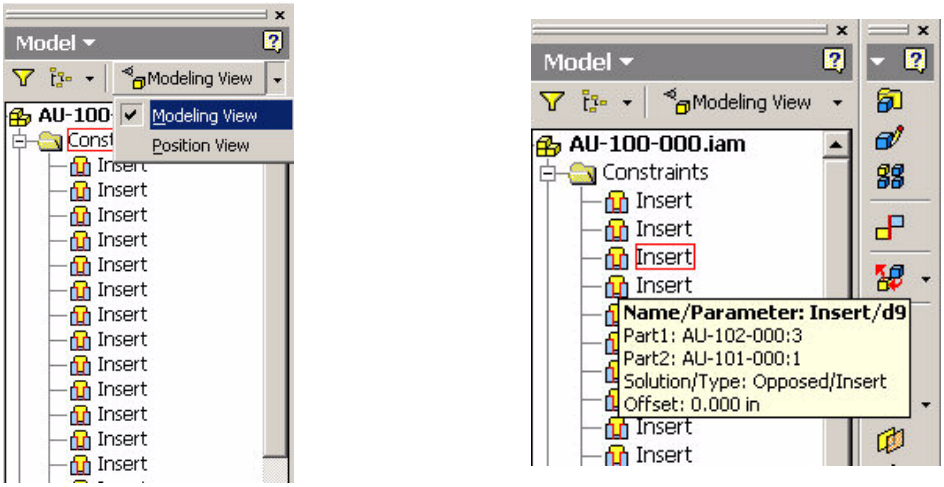

Figure 4 - Modeling vs. Position View

A tip that will help you prevent some errors is to delete redundant constraints when they are not used. For example if you place a angular constraint between two faces equal to 0° then later place a mate constraint between them, you will receive an error if you change the value of the angular constraint to anything other than 0°. Deleting the redundant mate constraint will help to eliminate problems. Redundant constraints can often been identified by the white (i) next in place of the icon. Critical constraint errors are identified by the yellow (!).

Figure 5 - Redundant vs. Critical Constraint Errors

The biggest single tip I can give about resolving bad constraints is this: practice. Nothing will help you more than using the program. When I first began using Inventor I had problems tracking down bad constraints too. Now, however (and I have seen other users develop this way as well) I can determine the root of a constraint failure in a few seconds.

10. Just Forget About Constraints

If after all of this you still find yourself banging your head against the wall when dealing with constraints then just forget about them. That's right, you don't need them. There is a technique called Skeletal Modeling (a.k.a. Master Sketch Modeling) that does not use modeling constraints. You build all parts off one master sketch and each part's position is dictated by the position in the sketch. You don't have to use one technique or the other exclusively either. Many users (including myself) use each technique for different modeling tasks.

For more information on Skeletal Modeling, see these web links:

http://www.sdotson.com/freetut/Introduction%20to%20Skeletal%20Modeling.zip

http://www.mymcad.com/KWiK/imaster/imaster.htm

http://tinyurl.com/a33n