The Skeleton Sketch Part (SSP) Introduction By - (John Stoltzfus)

1/27/2017 – REV A

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Skeleton Sketch Part (SSP) is or can be the same terminology as a "Master Sketch Part" –

I found this to be very helpful in the New Product Development process, because you can literally change out a part without disrupting any of the other components, however there will be mate errors showing if you were to mate another instance of the part within the assembly and those are easy to delete. Or change the shape of a part from round to square or the other way around..

You don't have to change how a part is modeled, let's say Extruded or Cut etc, this is not a document of how to create a surface helix or using the hole wizard.. It's a system with the ability to have a comprehensive parametric model, it's really not a new way to model, just a "New" thought process, to some, a different approach to design. There are just a few important steps that have to be followed if you want good robust models.

Do you enjoy designing with SolidWorks? I sure hope so.
Are you producing great designs that are trouble free? If you are, great carry on and don't change your system.

If you feel you would like to have robust parametric models then maybe you should review the following pages. If you're really interested, then study the following pages and try the process…

My Skeleton Sketch History –

The Neil Sardinas Way (**SolidWorks** Instructor and Applications Engineer at Prism Engineering)

In 2004 I had a one on one Advanced Assembly SolidWorks training with Neil Sardinas from Prism Engineering our VAR at that time and he showed me his model of an entire Steam Engine Train Car complete with the Coal Car, it was an impressive model, the largest SW Model I have ever seen. I recall he was working on the Coal Oil Lamps over that time. What really popped out to me was the entire train was done in separate zones and each zone had a matching sketch and was completely independent of the next zone, but each Zone was an integral part of the entire assembly. I forget how many zones he had it broken down to, I'm thinking dozens. The neat part was you could have had that many people working on the project, because everything was split and each split or connection point had the same connecting information, once the assemblies were dropped in a main assembly there were no interferences.

It took me awhile to totally adapt to his approach, my guess close to 5 years or more, and then I would try this, then that, as you know "in context" design everything worked well till you have to change a part from square to round, or delete a part, as you can imagine there was a mess to clean up and fix, the time it took was directly related to the placement of the part, was it the first part or the last part in the feature tree. I did modify his approach, or rather fine tuned it for my application, now I have no assembly that is done the old way, it is done the Neil way. I can delete any part within any assembly and not throw an error (except Mating Errors), unless I made a mistake, such as picking a vertex of a part where I thought was a vertex of a sketch, that is why I stress to "Isolate" the part/s that you need to edit.

IMPORTANT UPDATE – 3/30/2017
Assembly Template Structure

Do you understand how SolidWorks rebuilds the feature tree and all the components and how that can affect the assembly structure?
SolidWorks rebuilds from the top of the feature tree to the bottom and which is the main reason that you want the Skeleton Sketch Part as the first Item in the feature tree.

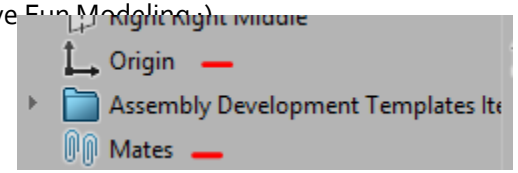
Just a recent find on the SW Forum, brought up by SW employee Frank Ruepp, (https://forum.solidworks.com/message/729015#comment-729015), (which was further explained by another SW employee Jeremy Regnerus in that same thread), was that when you add sketches in the Assembly it drops them down to the bottom of the feature tree and if by chance you would use the geometry of that sketch for any of the parts, guess what, really good chance you just created a Circular Rebuild, which is a data killer in SW.

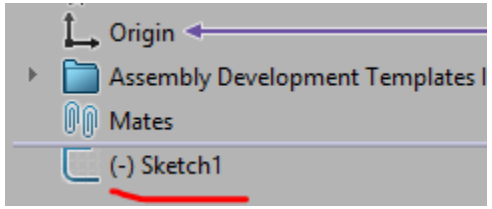
A sketch added after you insert a component goes to the bottom of the feature tree, so picture the sequence of events in a rebuild, SW starts with Part 1 and pages through all the sketches and features and suppose you had a sketch in Part 1 that is in context of the Assembly Sketch that's way down at the bottom of the feature tree, now SW can't give you a clean pass on that sketch and it hangs in limbo till SW reaches the bottom, now you should rebuild again, just to make sure Part 1 is clean. Now if you relate a lot of components that way, you can soon understand why things crash and burn…

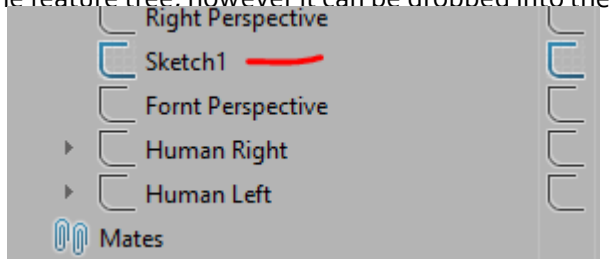
The great news is this:

Now you can create/update your Assembly Template add a sketch or sketches AND THEN – right click those sketches and add them into a folder and now the folder will be placed between the Point Of Origin and the Mates.. Now if you add a sketch it will be placed at the bottom of the feature tree, however it can be dropped into the folder, now a rebuild will be a lot cleaner and the updates will be a lot more positive..

Have Fun Modeling:)







Interesting enough – If you take that same sketch and use it in the Main Assembly for a Cut Extrude that you decide you need for a quick fix, guess what that feature and sketch just drop to the very last place in the feature tree, not the best and I will try to avoid it like a plague, I know some people like plagues, but I'm not one of them…

Top – Down – Bottom – Up (Notes/Speculation)

For me any position is good - Sometimes I just don't know what's up….

The main reason Top down approach doesn't work for a lot of people is the simple fact that there isn't a common denominator controlling it such as a Master Model or a Master Sketch, plus it can be way harder if it's not fully understood, it takes more thought up front, (Design Intent).

Starting your design the wrong way, (if you build top down or bottom up), can be just like you built a tower of blocks when you were a kid, as soon as you take out or try to change any of the blocks mid way or wherever in the tower, the crash will be the same, it takes work to rebuild it, most times people will do it the same way again and then complain that SW doesn't work, doing the same thing over and over doesn't get you different results.

The key in good top down design is using a Skeleton Sketch/Master Model approach and when you add a new part link it to the Master, that way if and when you pull out any of the blocks in the tower the rest stay put. Anytime you insert a new component the key is to Isolate the new part and the Master Part/Sketch, that way each component is independent but tied to a common structure..

So my point is either way you create your models, if your references are screwed up then your model is static and will blow up and be very difficult to change, I hear guys creating models and then deleting the references, I sure hate to edit those models as well.

One thing to keep in mind is you can reach up one level for information and drive that information down, if you try to reach up two levels for controlling information then the parts won't rebuild correctly, unless all of the controlling assemblies are open.

Any time you make any changes to your Templates, Save them, Change your Drafting Standards, Save them, Change your Tool/Options then use Copy Wizard to Copy your Settings – Save –SAVE-Save

Zone/Skeleton Sketch Advantages

- Each Zone shares the same geometry, therefore it doesn't matter which Zone you are working in, you can easily double click on any sketch or plane in the feature tree and have dimension pop up to change, without opening the part.
- Each Zone also shares the same Point Of Origin, which makes it a Snap to Assemble the project, drag and drop, reduces mates.
- Zero Interferences if you use the correct Planes and Sketches
- If there is a known change request, you can open only the SSP and change the known sketch, without having to open any assembly or part, then when you open the assembly the change will occur within the assembly and part files that would have been affected.
- You can edit the SSP in any of the different Zones/Sub-Assemblies, without opening up the SSP separately.
- When editing a component or adding a new component, it is good practice to "Isolate" the SSP and the part being added/edited, that way you don't inadvertently select a vertex, line or a face of anther component, or if you need to edit the SSP, it is also good practice to "Isolate" as well.
- Parametric Changes, change a dimension and watch the part move instantly when the part is rebuilt (Ctrl Q)
- Possible to stack or have multiple SSP's, handling different Zones, however, if there are more then one, they should be in every sub-assembly as well, to assure that the proper connection points are met.
- You can Cut/Boss Extrude up to planes or vertexes, this assures no direct link from part to part, which can be a huge advantage in any "New" design processes, because with no direct link when you delete a part in the feature tree there are no "Errors" from one part to the other.
- Changing a part from Square to Round is just a small operation when following these instructions, change the sketch in the SSP and open the effected part, open the initial sketch, delete the square and convert the entities of the circle and your back in business..
- Adding Design Tables and Configurations
- The SSP doesn't need to be "Just" sketches or planes – you can just as easy use surfaces or solids if that makes your life easier.
- Also you can name your SSP a Master Sketch or a Master Model Sketch, it doesn't really matter what you call it, everything you do in SolidWorks starts with a Sketch, so its easier just saying SSP rather then a Master Model Sketch, or Master Model, that's takes much more energy saying all those syllables, SSP, that flows :~} {~:
- For easy changes "Name" your sketches and "Name" your sketch dimensions that drive the model, then you can go to the **SSP equations folder** and filter all the dimensions and make the changes there as well, that's a cool tip off the Forum (see the Tricks thread).

Zone/Skeleton Sketch Disadvantages

- **(this page is mostly blank) for you to fill out and send back, if and when issues occur. Good chance that I'll probably write back and tell you did something wrong, sigh)…**
- None known to exist – worse than a endangered species

Skeleton Sketch Will Eliminate

- Continuing gray hair or bald spots (if those symptoms are caused by screwed up SW models)
- Errors if you need to delete an item in the feature tree, (it will cause mate errors if another instance of the part is mated in the assembly etc)
- Feature Tree Forrest Fire (Smokey the Bear is headed for retirement, it's about time, he's really really really old)
- Plus a lot lot lot more……

Skeleton Sketch Assembly Hierarchy

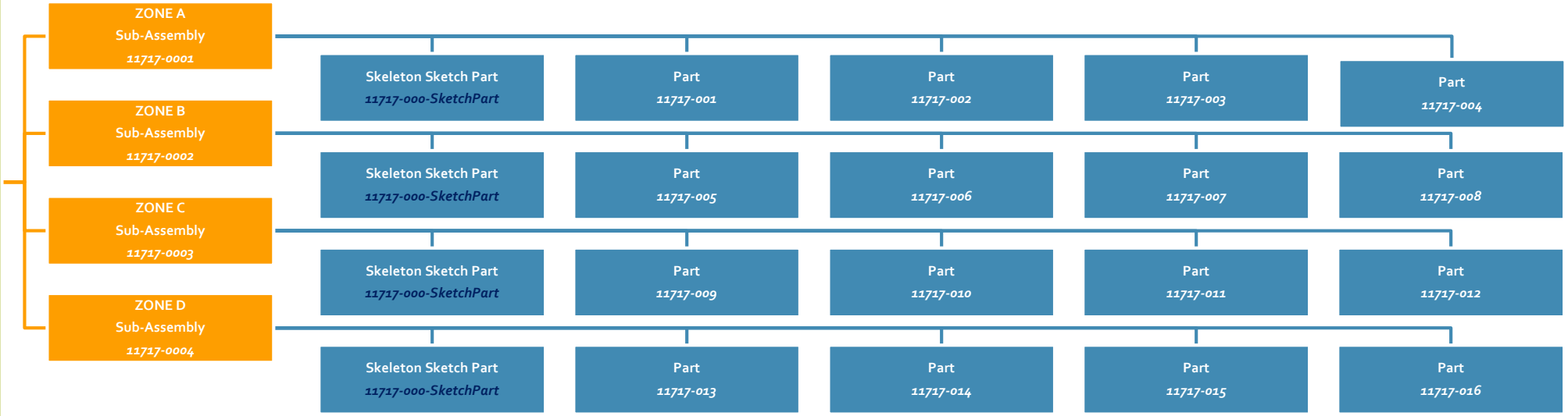
TOP ZONE

Main Assembly

11717-0000

Also Includes Part #

11717-000-Sketch Part



Feature Tree Example of a Small Bench

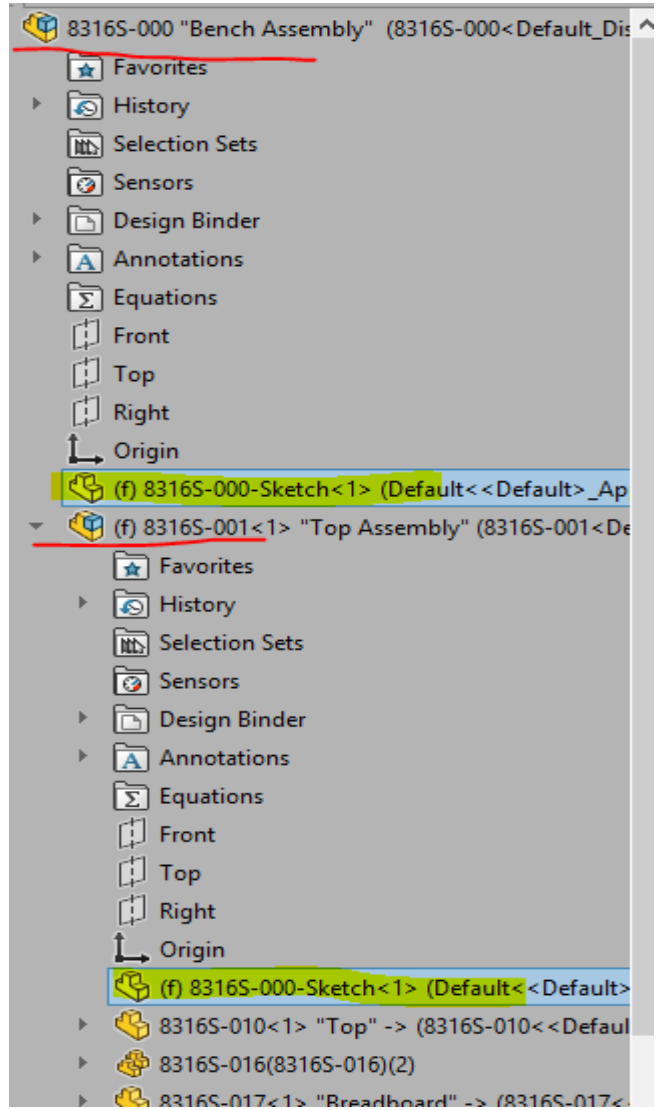
The feature tree shown has only (4) four sub-assemblies which consists of (1) one sub-assembly for the top, (2) two sub-assemblies for the legs and (1) one sub-assembly for the center stretcher assembly..

8316S-000 is the Main Assembly Number and the first item listed in the feature tree is the SSP with the same number as the Main Assembly with a suffix of –Sketch, (8316S-000-Sketch).

8316S-001 is the Sub-Assembly for the Top or Bench Seat, and the first part in the feature is part number (8316S-000-Sketch), which is the same

8316S-002 & 8316S-003 have the same part, (8316S-000-Sketch) in the feature tree..

Parts that get connected at manufacturing and stay connected through finish and packaging are treated as a Zone/Sub-Assembly.

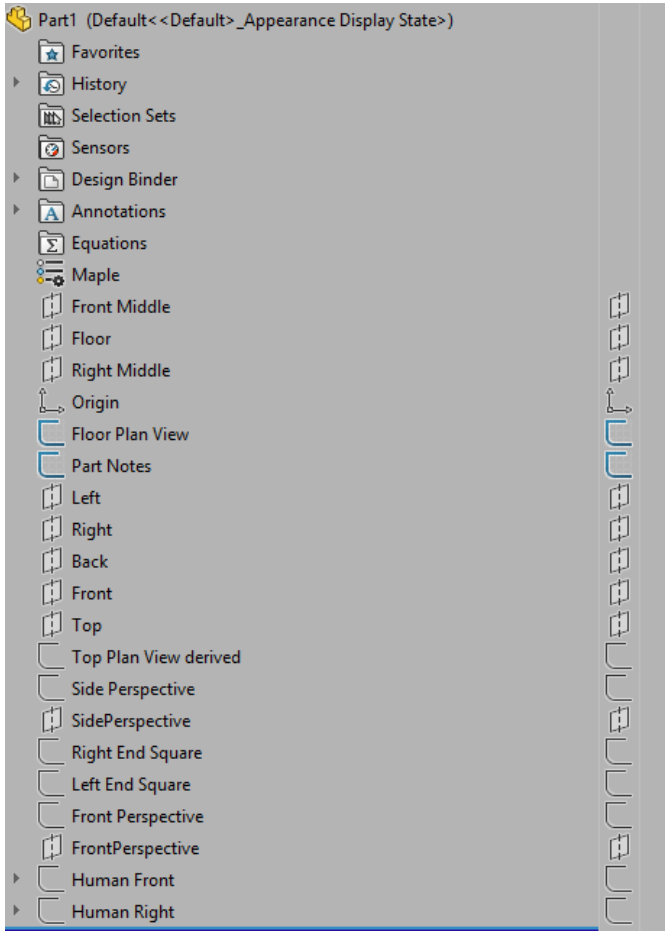


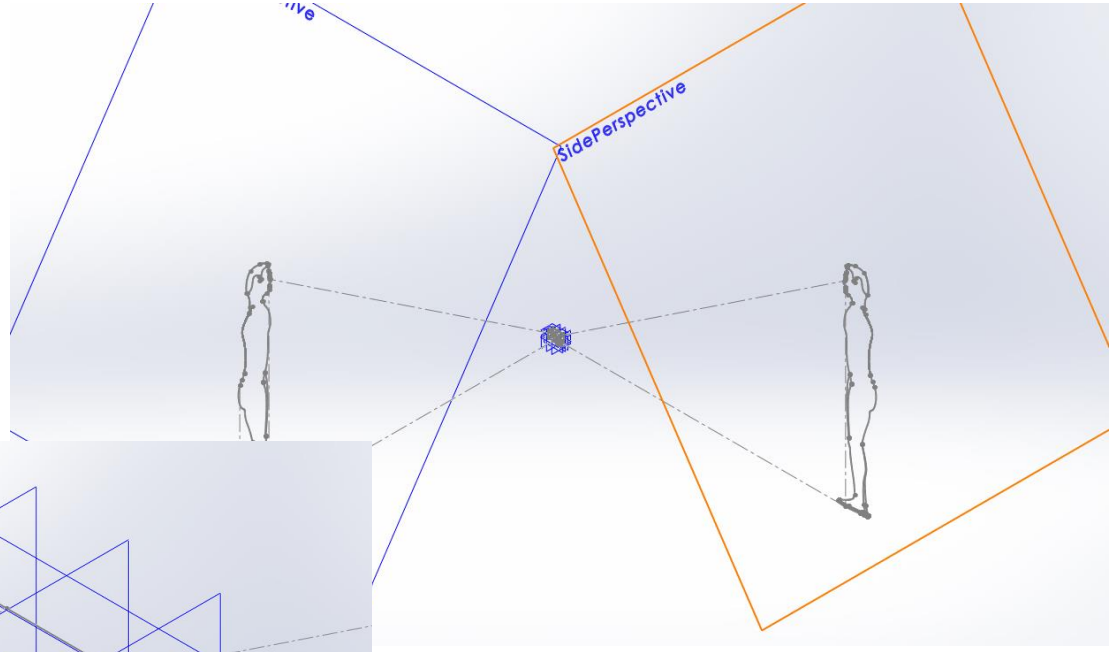
List of Items In My "New Part" Template

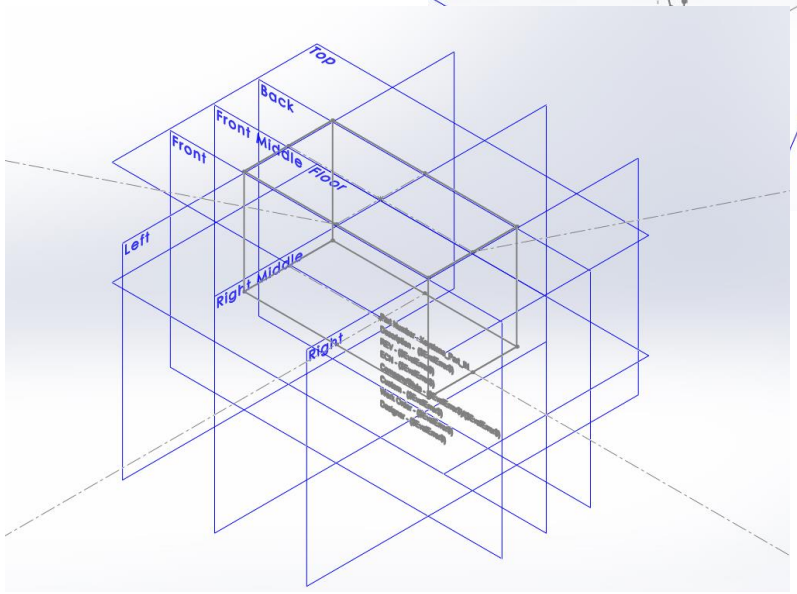
1. I changed the names of my Planes – Top plane is now "Floor" – Right Plane is now "Right Middle" – Front Plane is now "front Middle"
2. 1st Sketch is a rectangle sketch attached to the Floor Plane and is named Floor Plan View
3. My 2nd Sketch is called Part Notes attached to the Front Middle plane and only consists of Text (Tools/Sketch Entities/Text) and the Text is a few Project Custom Properties, this sketch is almost always hidden, but there for a quick reference.
4. My 1st Plane is the "Left Plane" and the references are the Right Middle Plane and a vertex of the left side of the Floor Plan View
5. The 2nd Plane is the "Right Plane" and the references are the Right Middle and a vertex of the right side of the Floor Plan View
6. The 3rd Plane is the "Back Plane" and the references are the Front Middle and a vertex of the back side of the Floor Plan View
7. The 4th Plane is the "Front Plane" and the references are the Front Middle and a vertex of the front side of the Floor Plan View
8. The 5th Plane is the "Top Plane" which is offset from the Floor Plane
9. The 3rd Sketch is "Top Plan View" which is derived from the Floor Plan View and the Floor Plane
10. The 4th Sketch is the "Side Perspective" which is only 3 lines one horizontal, one vertical and one diagonal, the diagonal line is for a line of sight and used for new designs
11. The 6th Plane is Side Perspective" and the references are the diagonal line end point and the diagonal line (we select this plane and go normal to, this gives us a perspective from the Line of Sight.
12. The 5th Sketch is the "Right End Square" and the references are the Right Plane and the Top and Floor Plan View Sketch corner vertexes.
13. The 6th Sketch is the "Left End Square" and the references are the Left Plane and the Top and Floor Plan View Sketch corner vertexes.
14. The 7th Sketch is the "Front Perspective" which is similar to #10 with the except the Plane used #14 uses the Front Middle Plane
15. The 7th Plane is "Front Perspective" and uses the diagonal line end point and the diagonal line as the references
16. The last 2 sketches are only a Block of a Life Size Human to give us the size perspective plus the diagonal line of sight height etc..
17. Yes there is a lot of stuff inserted – but really nice when starting a new project, first I double click on the Floor Plan View sketch and change the dimensions and double click on the Top Plan and adjust the height, now my overall's are established..
18. I have also inserted a Sketch with text for a Memorization Chart and a Decimal Conversion Chart - See page 12

My "New" Part Template

Yeah a lot of stuff – but this is where I start





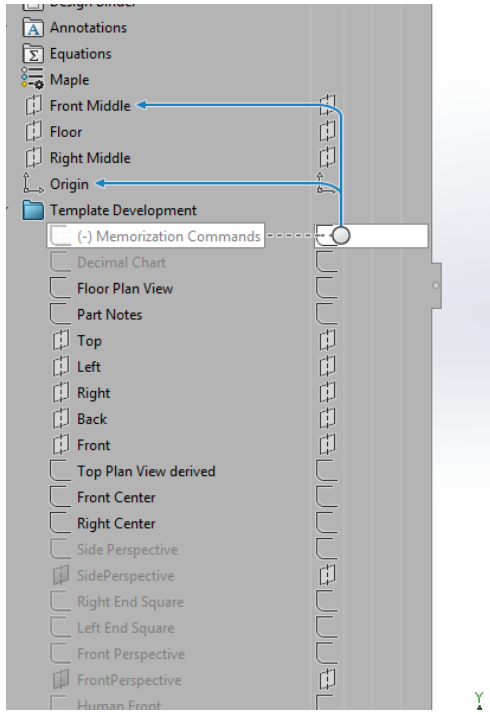


My "New" Part Template…. Continued

We keep learning everyday if we keep looking and searching… Just recently Jaja Jojo posted on the forum (https://forum.solidworks.com/thread/183132?start=405&tstart=0 Page 28) - showing the different commands in the background as a memorization chart, wow..

What I did is updated my Part and Assembly templates by inserting the chart in the Front Plane as a Text Sketch – The nice thing about this is you can suppress the sketch so it doesn't bog down the system or the file, so all you need to do is hover your mouse pointer over the sketch in the feature tree and it will show.

This was one of the best tips that I'd seen in a long time.



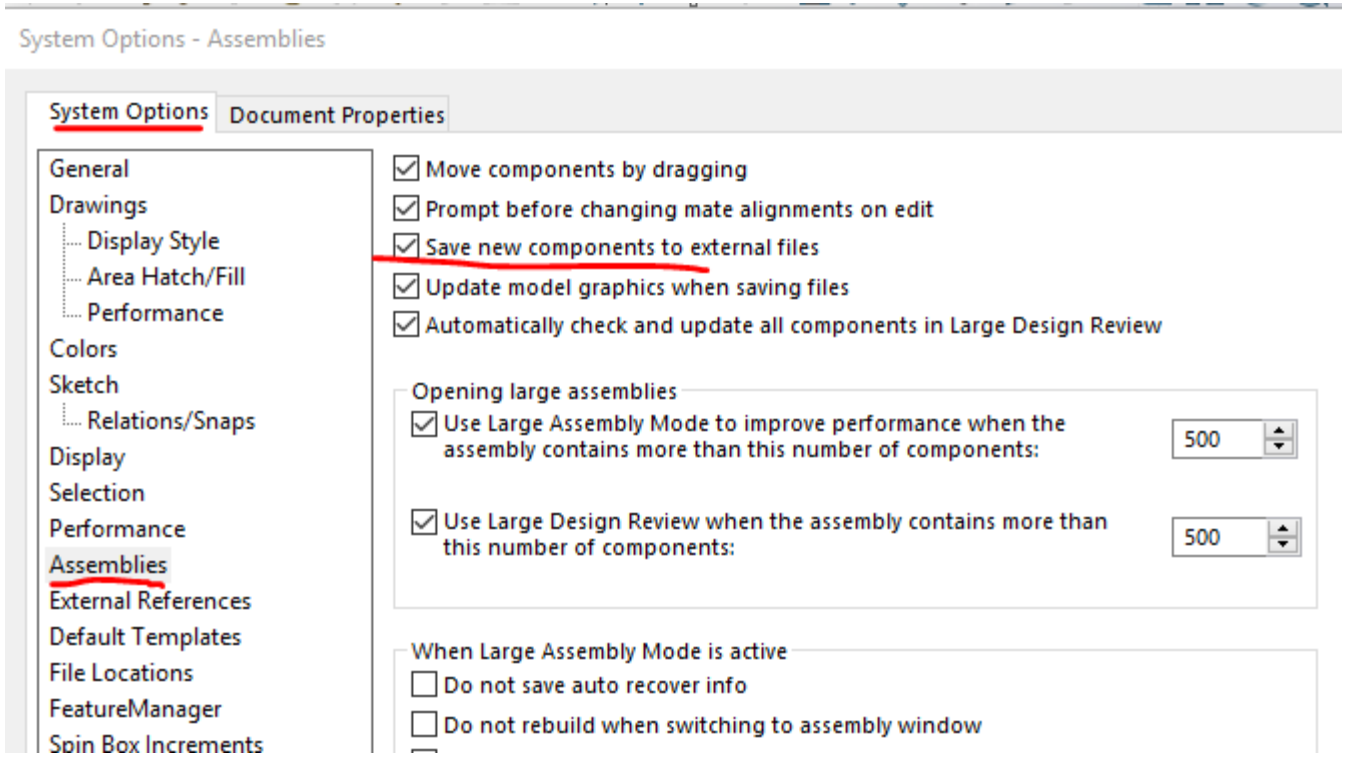
How I Apply The SSP

1. Open a "New" part and save it as the SSP.
2. Figure out approximately how many Sub-Assemblies I need
3. Open a "New" Assembly and save it (you will need one assembly to represent your Main Assembly)
4. Insert the SSP in the just created Assembly file, by hovering over the point of origin and when you see the Double Arrow let it drop
5. Save
6. Then do a Save As Copy for however many Sub-Assemblies you think would be required for the entire project, this don't have to be accurate, you can always delete any Sub-Assembly you don't need or add another one, if you add one follow #3 and #4
7. Now open the Main Assembly
8. Insert all the just created Sub-Assemblies, use the same method as in #4
9. Save and close
10. Open the SSP
11. Apply the overall dimensions (Floor Plan View & Top)
12. Add known Planes/Sketches and use those sketches to build the design. (Tip – Name the Planes/Sketches, so they're easy to find) – (Tip – try to keep similar Zone Items together so they can be dropped in a folder, this takes away the feature tree confusion syndrome)
13. You can add surfaces as well as solid extrusions if it helps in the process
14. Add planes or sketch lines to use the option "Extrude Up To" when you do the solid modeling, this eliminates any connection to any of the other components, incase you would need to delete the part..

Check Your Settings Before Inserting New Component

(Updated 3/31/2017)

1. Go to Tools/Settings/Assemblies – Check the box that reads - "Save New Components to External Files"



Having this unchecked will prompt to save as a virtual file, ok it that is what you want…

Inserting A New Part

1. Open a Sub-Assembly where you want the part to be built
2. Go – Insert/New Item (have your Tools/Options set that when you insert a new part it prompts you to save the part file right away)
3. Save the Part file
4. Then SW will wait till you pick a plane to drop the part into – "ALWAYS" pick the Sub-Assembly Front Plane which would be the "Front Middle" – don't draw anything
5. "ALWAYS" Close that sketch
6. Select a plane where you want the part, which would be in relation to one of the sketches in the SSP
7. Insert "New" Sketch
8. Select the proper sketch from the SSP and convert entities.
9. Extrude up to either a vertex or a plane
10. This is where it pays to spend a little more time with the sketches and planes in the SSP, it may take me a day setting one up, but with an hour or so my assemblies are complete and ready for review, as you know that really depends on the number of pcs and the complexity of the design.
11. "ALWAYS" When you insert any subsequent components always select the SSP and the New Part in the feature tree and right click and "Isolate" – doing this is what allows you to delete any item in the feature tree and not kick errors.
12. For kicks and giggles close the Sub-Assembly your working on and open the Main Assembly, for more giggles, open up a different Sub-Assembly and add a few parts now go back to the Main Assembly, soon you be ROTFLYBO
13. It's not "New" exciting tools, again it's a systematic method that creates solid designs every time.
14. Pack and Go is so easy if every part is changing and there are no interchangeable components, just change the Floor Plan and Top Plane dimensions and everything changes accordingly..

Did You Know

- That - You can Color Your Sketches – making them easier to find?
- That - You can add Configurations to the SSP
- That - SSP eliminates Circular Rebuilds
- That - Creating Parts and Assemblies with SSP – you can reduce the number of Mates, by dragging and dropping the part or assembly onto the point of Origin
- That - Designing/Modeling is only half the work in getting the information to manufacturing or to potential clients and Custom Properties and Drawings have been evolving for me over the last few years. The introduction of Custom Property Tab Builder had been an awesome boost when it comes to getting the information consistent, easy to change etc…
- That - I work with multiple tab drawings, (some over a 100 drawing sheet pages),
- and never ever ever want to go back to single drawing files or drawing file per part. Using that many drawing tabs can slow down the file, however it is still quicker than opening and closing a 100 files. My drawing setup does have the ability to have individual part or individual assembly information within the part file, you change one part, the CPTB is also setup to handle up to 3 revisions, so the information is right where it needs to be, with that part or assembly. Any interchangeable component is listed in the BOM only and there is a drawing in some folder somewhere within the system.
- I have well over a 180 custom property fields that can be used, naturally I don't use nearly all of them, but I have the flexibility to use them when I need to. Each of the Custom Properties were needed at some time or the other, so I would just keep them in the CPTB file. Within that CPTB file I have room for 60 notes and most times I might only use 3 or 4 lines per part, one assembly had over 40 lines of notes, so I just added a few more.
- That -the New "Visualize" does some awesome renderings
- That - the SW Forum is a good place to get help (Mental Doctors are on call 24 -7) – they will always tell you not to jump……
- That – Years ago I uploaded a lot of SolidWorks Models to www.3dcontentcentral.com – some may be good and some may not be so good, your choice, but you can go there to download files, if you need Motors, CEMA equipment, Hardware plus some other stuff.
- You can connect via www.linkedin.com – if you know how..
- My private personal semi secure email, just recently hacked, Yahoo account is jjffiigg@yahoo.com, if you want to send me stuff you wouldn't want to show your mother… (and we're pushing for Cloud Storage, what up wid dat)

COMPLICATED SELF INFLICTED DESIGN
2016 UI Debacle

WARNING:

Don't post this on the SW Forum – In retaliation of the New 2016 UI

(like I did and almost got a life ban as the moderator spent a few hours on Google searching for Randolph Van Henzel from the AP – in Volgenstruas DE, to try and find more info…)

Randolph Van Henzel

VOLGENSTRUAS, DE – AP - Seems a German Design Engineer's eye strain got the best of him on Dec 26th 2015. Designing new products can be a headache just by itself, it's only gets worse after using a Design software that has just recently released its new UI for 2016.

Apparently, Alfred Strassenhaus of Volgenstraus got in a little bit of trouble and a lot of hurt, after working 12 hours behind a computer, while under pressure of a looming deadline.

According to the local Chief of Police Willem Martens, Alfred went home with a pounding headache so he took a few too many paracetamol and downed them with some homemade brew. Soon afterwards he went for a walk in the park and while there he became incoherent, and a little unruly. Unfortunate for poor Alfred and his trying to smooch his way into the arms of a women, who by the way, is married to the towns past best high school wrestler, (even after being out of wrestling for over 15 years), poor Alfred had no chance. While in the process of being plummeted, he finally breaks free and runs as fast as possible, which wasn't really fast, but fast enough that when he went over a railing and landed on a stairway he had enough of speed to roll all the way down, about 10 flights of concrete steps, which is what accounts for all the abrasions to the head and upper body. After laying in a prone position for some time he finally tries to walk home, somebody should have told that going south out of town isn't the best idea either. Soon he left the asphalt and kept on walking till it seemed like the ground gave way and he fell or rather tumbled down an embankment right in to a partly frozen stream, which accounts for the gash across his face. With blood running down his face poor Alfred looked like a horror movie actor, he finally crawls up the bank and starts walking again, somebody should have told him, "don't play in traffic". When he was in the process of crossing a small back road a small black BMW came around the turn and caught Alfred by the pants which completely tears them from his body and breaks both legs which is why he is in a body cast up to his waist. Additionally preying charges and costs to fix the BWM will pinch him later. According to the family he will recover, but plans to buy another Design software.

Furniture Design