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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 just as 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, 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...
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, I would try this, than 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.
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 another 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.
**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)
- Plus a lot lot lot more......
Skeleton Sketch Assembly Hierarchy

TOP ZONE
Main Assembly
11717-0000
Also Includes Part # 11717-000-Sketch Part

ZONE A
Sub-Assembly 11717-0001
Skeleton Sketch Part 11717-000-SketchPart
Part 11737-001
Part 11737-002
Part 11737-003
Part 11737-004

ZONE B
Sub-Assembly 11717-0002
Skeleton Sketch Part 11717-000-SketchPart
Part 11737-005
Part 11737-006
Part 11737-007
Part 11737-008

ZONE C
Sub-Assembly 11717-0003
Skeleton Sketch Part 11717-000-SketchPart
Part 11737-009
Part 11737-010
Part 11737-011
Part 11737-012

ZONE D
Sub-Assembly 11717-0004
Skeleton Sketch Part 11717-000-SketchPart
Part 11737-013
Part 11737-014
Part 11737-015
Part 11737-016
The feature tree on the 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 SidePerspective“ 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..
My “New” Part Template

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