

SOLIDWORKS CAD Best Practice Guide

**Suggestions and best practices for
working with SOLIDWORKS CAD.**

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Introduction

This guide is intended to be a universal guide for users who use SOLIDWORKS CAD. It outlines a collection of proven best practices designed to support efficient, accurate, and robust design workflows. By following these guidelines, users can reduce errors, improve model performance, and maintain consistency across projects.

Not all techniques will apply to your design environment but if you take the time to read and understand these guidelines, you can find which work best for you and your designs.

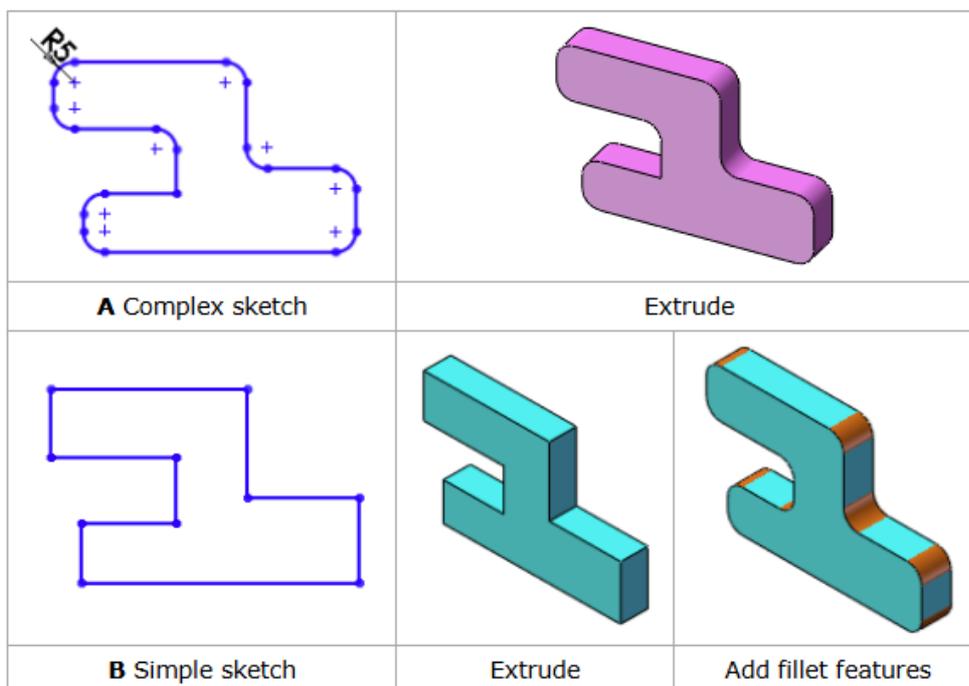
Sketching

In SOLIDWORKS, a sketch is a 2D drawing that serves as the foundation for creating 3D models. It's usually the first step in building most parts. They can define the size, shape or placement of 3D geometry and sometimes can be used for reference purposes. Understanding how to optimize sketches is crucial for producing accurate and reliable designs in SOLIDWORKS.

The following points describe some useful techniques for using and managing sketching.

Simple Sketches

Keep sketches simple and build up the model using features. Simple sketches are more flexible and easier to manage. ***This makes it easier to edit models.***

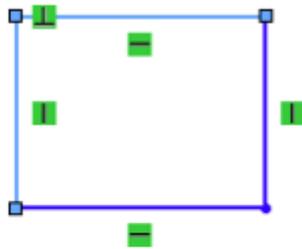


Design Intent

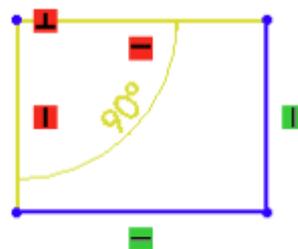
Design intent is important in SOLIDWORKS because it ensures that your model behaves predictably and intelligently when changes are made.

To achieve this, it is important to use relations and dimensions in sketches to capture the intent of the design. Relations help to identify the relationship between sketch items and dimensions identify their size. Some key recommendations are:

- Remember that relations can often eliminate the need for some dimensions, and they make your model smarter.

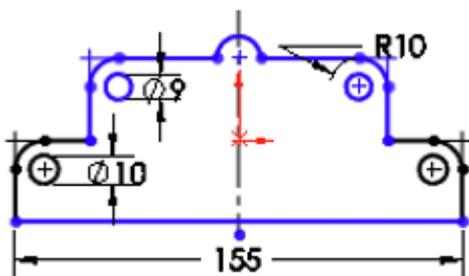


Relations are added to control the relationships between sketch items.

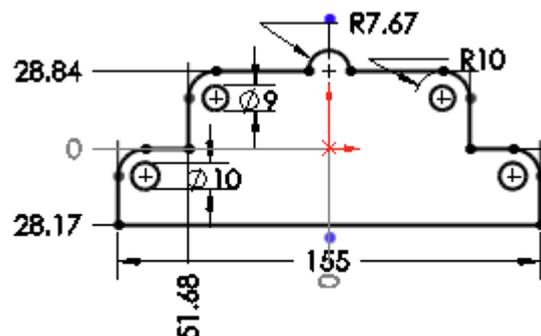


A redundant dimension is added. The sketch is over defined.

- Use symmetry and sketch mirroring where possible. ***This reduces the number of dimensions and relations needed.***
- It is recommended that all sketches are fully defined using a combination of relations and dimensions. ***This makes it easier to predict how the profile will behave when parameters might change.***



Under Defined Sketch



Fully Defined Sketch

Sketch Fillets

Fillets should be applied mostly as 3D features, and only as sketch fillets where needed. ***This allows fillets to be edited and/or suppressed as needed as most fillets are added later in the design.***

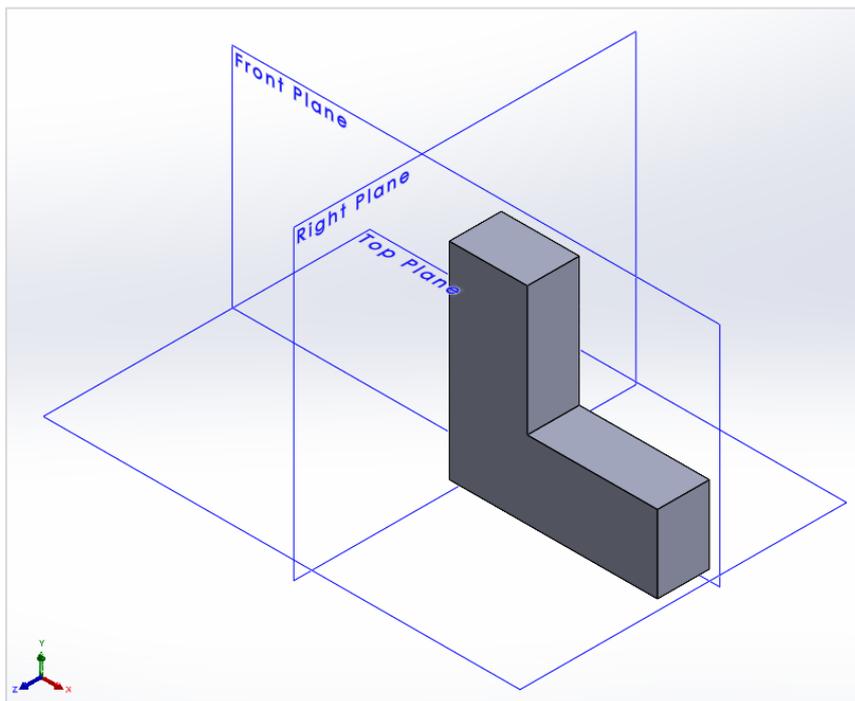
Sketch Options

- Switch on 'Enable on screen numeric input' for sketches. ***This makes it easier to add/update dimensions as you sketch.***
- To help visualise if a sketch is closed, you can switch on "Shaded Sketch Contours", which will shade closed loops in a sketch.

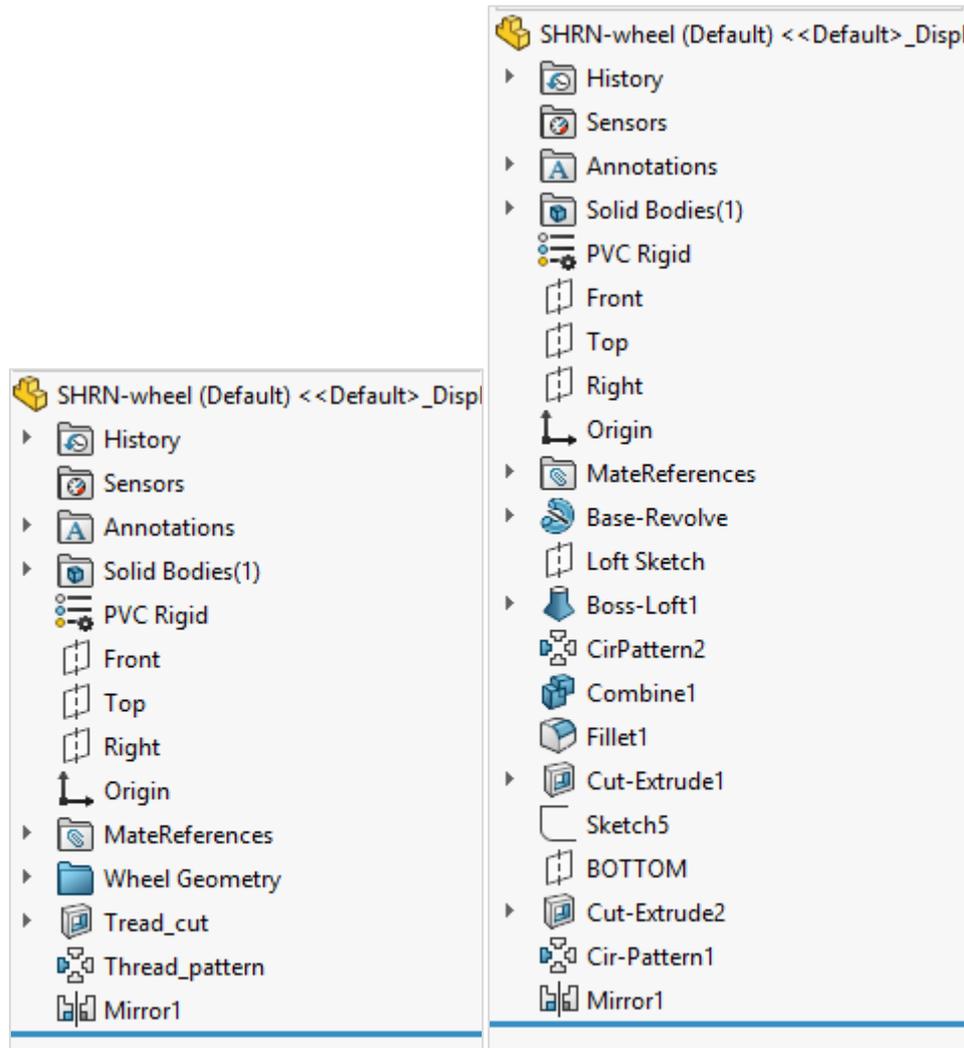
Part Modelling

In SOLIDWORKS, a part is usually a single 3D component built from 2D sketches with applied features. Effective part modelling relies on a combination of planning and execution. Following the best practices below will help create cleaner, more reliable models and save time during edits and downstream operations.

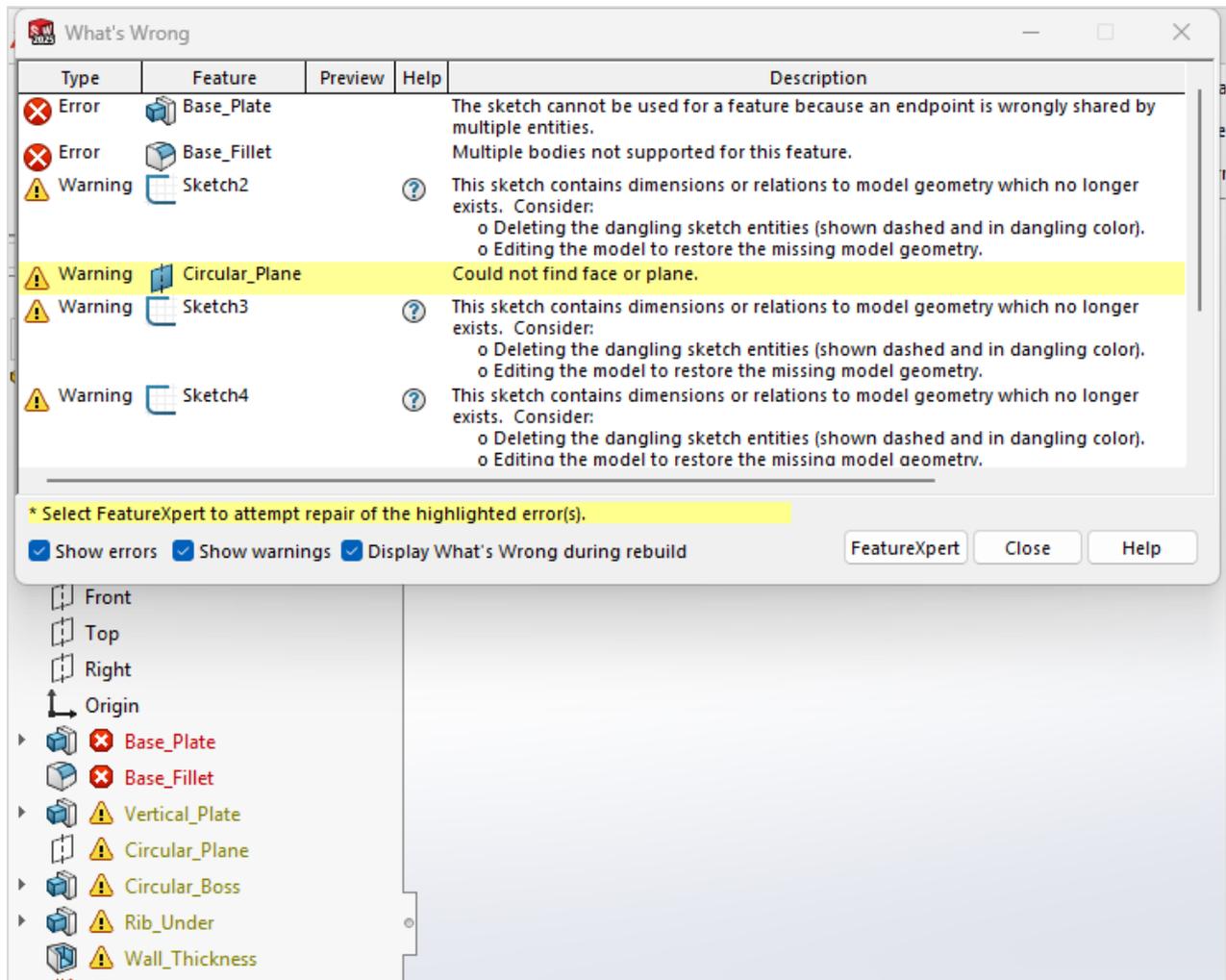
1. Understand the design intent and model accordingly. ***Having a well thought out design plan makes for a more robust design and facilitates easier editing.***
2. Orient the overall model in X/Y/Z according to the design intent. ***Orienting the model helps to see it as used in the real world.***



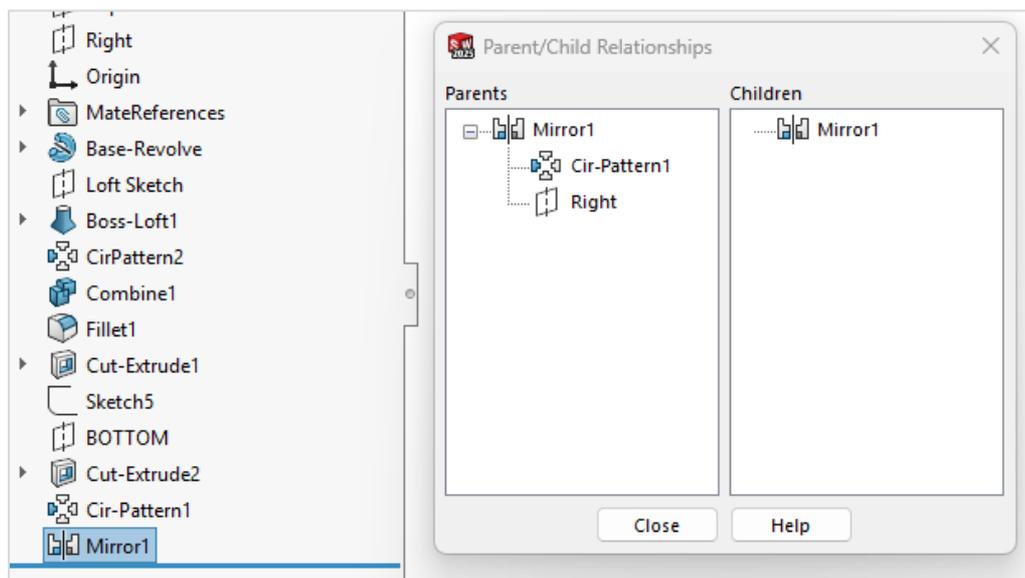
3. Name features where possible to capture design intent [Tools > Options > Name feature upon creation]. ***This will help you and others understand/navigate the geometry (or design intent) later or in design reviews. You will thank yourself later.***



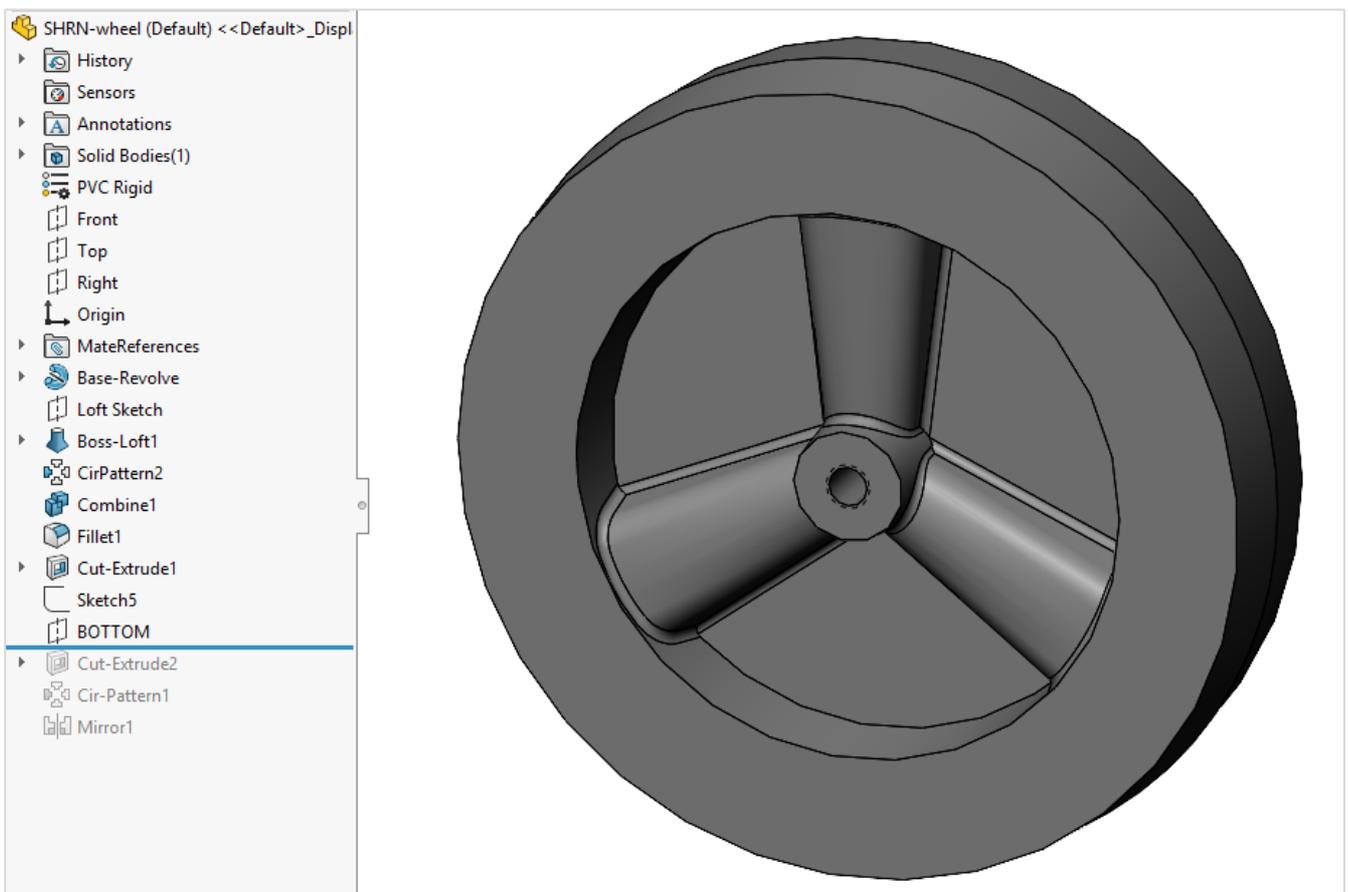
4. Don't model helical features for threads on fasteners: use cosmetic threads. ***This will provide best performance and allow threaded features to show the correct convention in drawings.***
5. Add cosmetic fillets and chamfer features towards the end of the feature tree.
6. Do not leave features in parts with sketch or feature errors. ***Errors cause distraction, confusion, and can cause other features to fail.***
7. Do not leave parts in rolled back state. ***Leaving parts in a partially rebuilt state can cause mate errors in assemblies and confusion for other users.***



- Limit in-context relations in sketches when designing parts in assemblies. ***This will reduce the dependence of the part upon the assembly it lives in and will improve rebuild times.***



9. Apply a material to parts. ***This allows for mass calculations, plus part templates with materials save time.***
10. Utilise patterns and mirroring where possible. ***Patterning/mirroring reduces rebuild time and simplifies editing down when changes are required later.***
11. Utilise symmetry and the Mirror Body feature where possible. ***Locating a part symmetrically with respect to the origin will pay off with simpler downstream features, mating, and drawings.***
12. Use Hole Wizard as opposed to Extruded cuts. ***The Hole Wizard embeds information about fasteners into the part for use in assemblies and drawing hole callouts.***
13. Advanced Tip: Use Library Features for common cuts or groups of features. ***If you find yourself making complex common features repeatedly, a Library Feature will pay off in reduced design time and consistent feature definition.***
14. Use the Rollback Bar and Freeze Bar to limit rebuild times on editing large parts. ***Temporarily suspending the amount of rebuild calculation can speed up design work.***

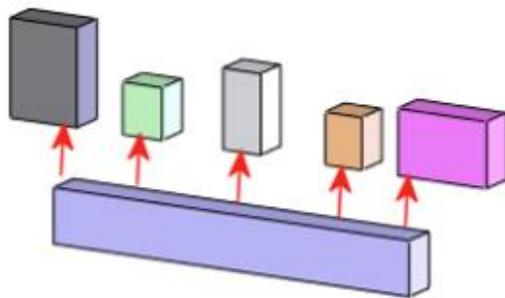


Assembly Modelling

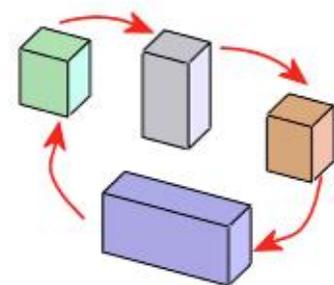
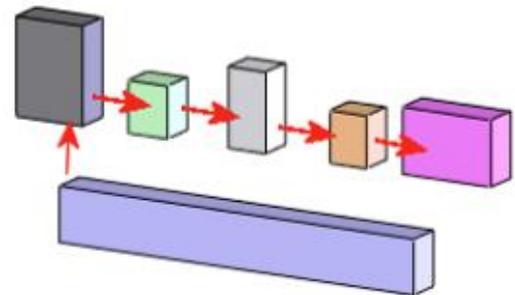
Assemblies in SOLIDWORKS are built from multiple components, which can be parts or other assemblies (subassemblies). Each component is positioned together using constraint mates or mates designed to represent mechanical relationships.

Effective assembly modelling can significantly enhance your productivity and efficiency. The best practices below will outline some useful guidelines for user on how to model and use assemblies in the most efficient way.

1. Use the least restrictive mate to do the job. **Less restrictive mates make over constraining the assembly less likely.**
2. Whenever possible, mate components to one or two defined components or references. **Long chains of components take longer to solve and are more prone to mate errors. (Mate Parts B, C, & D to Part A. Not Part D to C, C to B, B to A).**



A good mate scheme

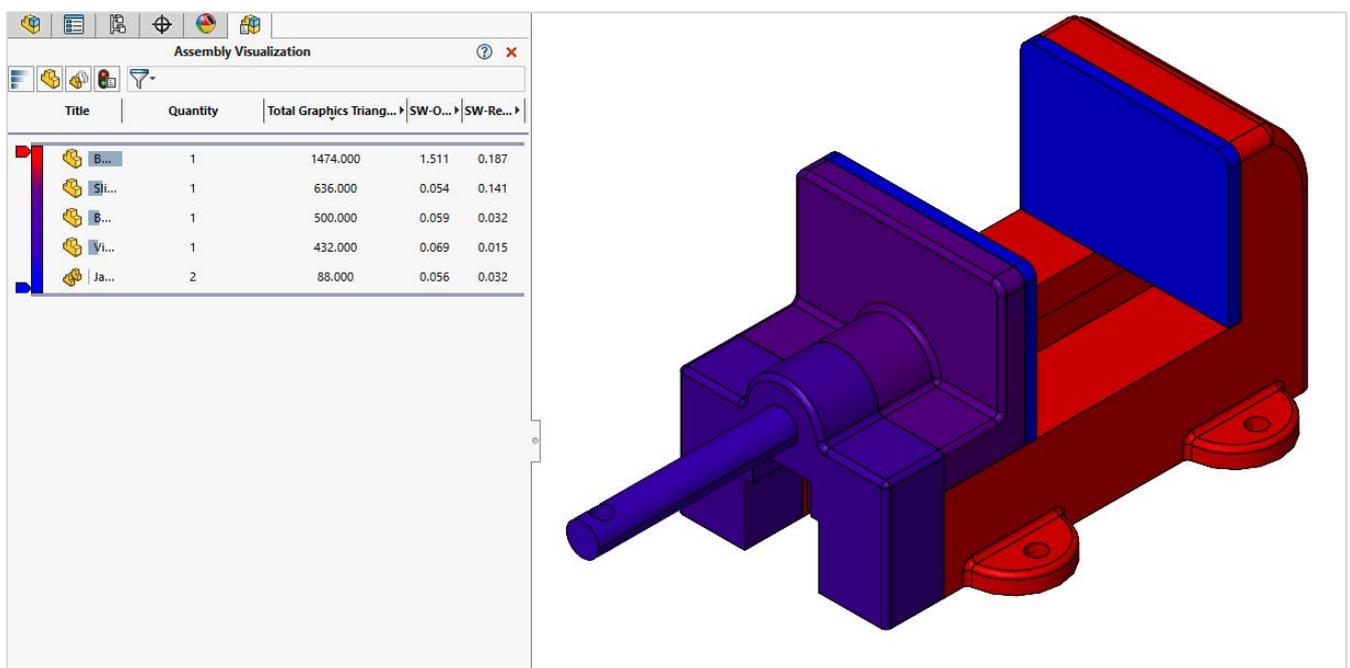


Mate schemes to avoid

3. Do not leave components in assemblies under-constrained or with mate errors. **This can lead to longer load/rebuild times for assemblies.**
4. Use option to Lock Rotation of concentric mates to Toolbox components. **This little-known option can resolve worrying minus signs in your component tree.**

5. Use component patterns where possible. ***This reduces computational load, reduces the number of mates, and speeds up rebuild times.***
6. Whenever possible, use subassemblies to reduce top level mates and hence the rebuild time of the top-level assembly. ***Flat structures with many top-level mates are computationally intensive.***
7. Do not use Flexible subassemblies as a modelling technique; use them more of a validation/motion technique. Avoid saving with subassemblies in the flexible state. ***Mating to flexible subassemblies is not allowed and making already mated subassemblies flexible can invalidate/over-constrain mates in the top-level assembly. Use care with this function.***
8. When using Toolbox, ensure it is located on a server so everyone can access and used the same shared data. ***This ensures that any added part configurations are available to all company users.***
9. When working with large assemblies use Light Weight & Large Design Review where necessary. Utilise simplified configurations. ***These are good practices to improve large assembly performance.***
10. Use Display States to save an assembly in different combinations of display settings (hide/show state, transparency, display mode, colour, and texture). ***Changing the visibility does not affect the rebuild or evaluation speed. Display performance improves, however.***

Tip: make use of the Assembly Visualisation tool to evaluate the performance of your assembly. To open this tab: Click *Assembly Visualization* (Tools toolbar or Evaluate tab on the CommandManager) or *Tools > Evaluate > Assembly Visualization*.



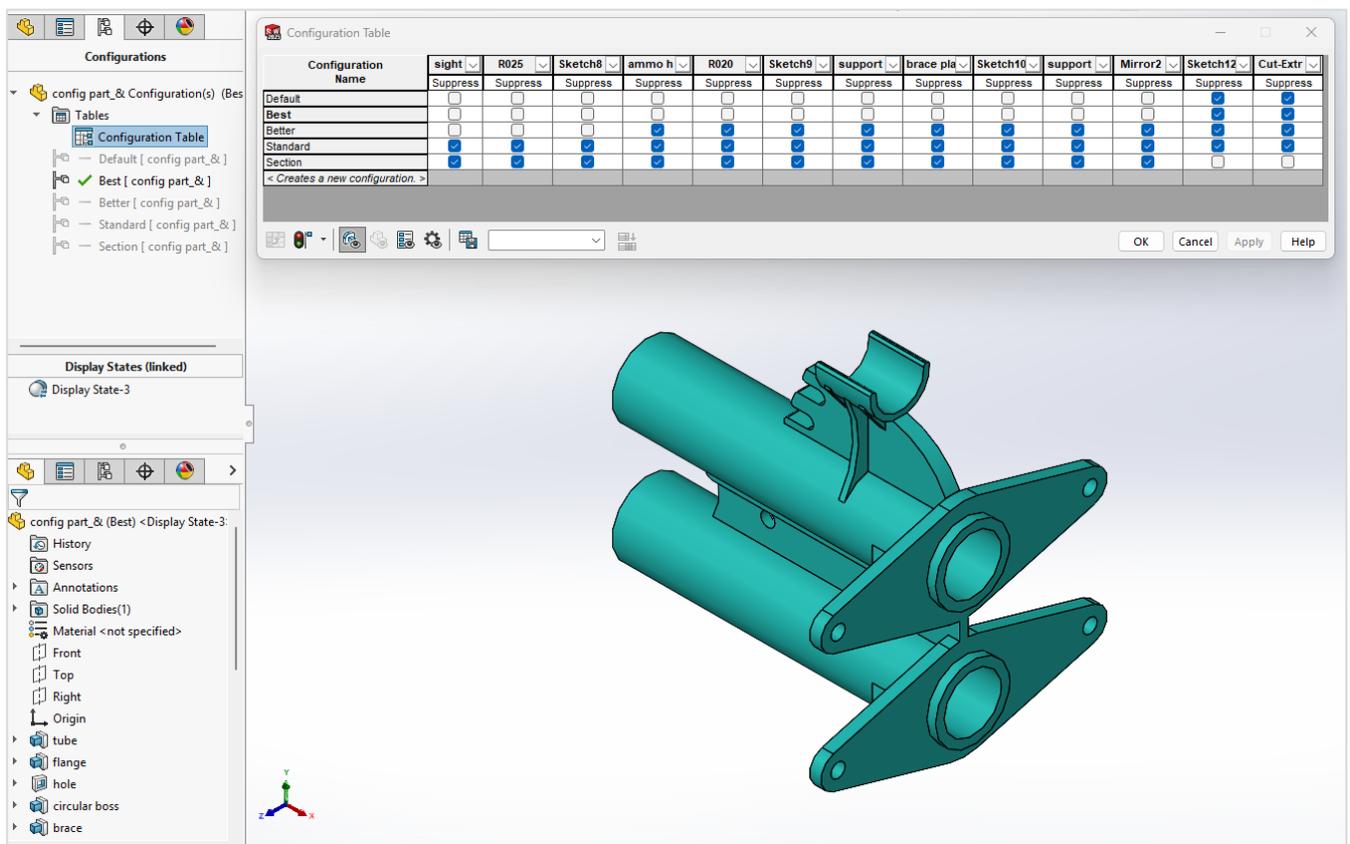
Configurations

Configurations allow you to create multiple variations of a part or assembly model within a single document. Configurations provide a convenient way to develop and manage families of models with different dimensions, components, or other parameters.

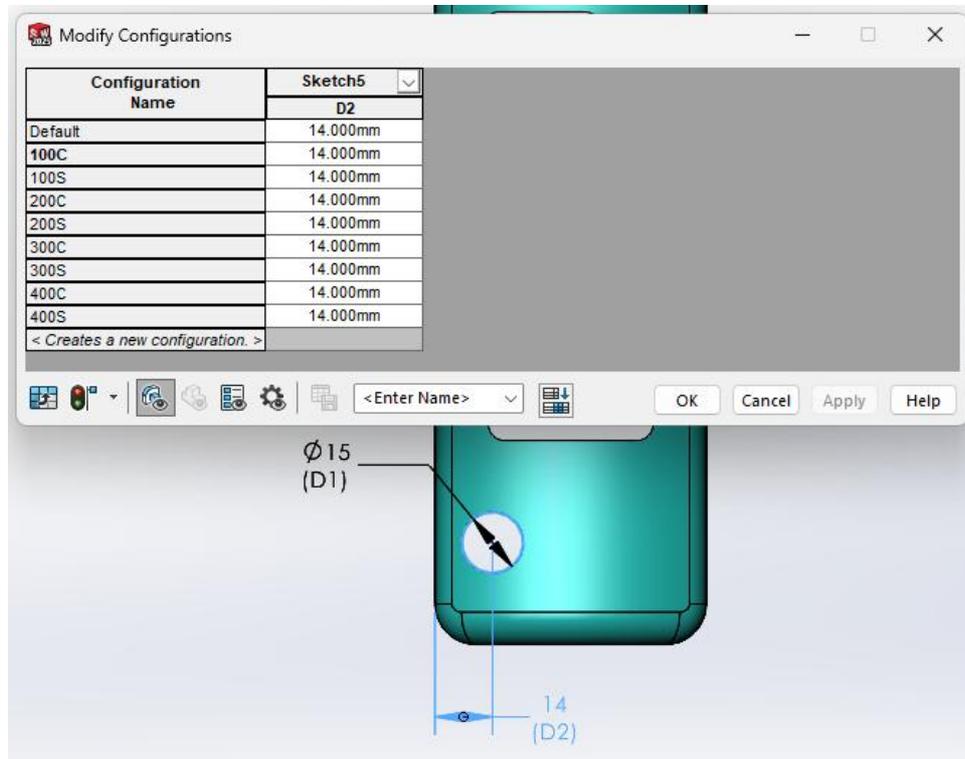
- Pros:
 - Would allow the CAD user to manage all product variants in 1 file.
 - Would allow a non-CAD user to view 3D of all product variants from 1 file.
- Cons:
 - Potential complexity of managing multiple configurations.
 - Would not allow different revisions of each of product variants in PDM (files are revised, not configurations).

Part Configurations

- You can configure a feature or features by selecting them, right clicking, and choosing *Configure Feature*. The resulting table enables you to make configs on the fly and control the suppression state of each feature in each configuration.



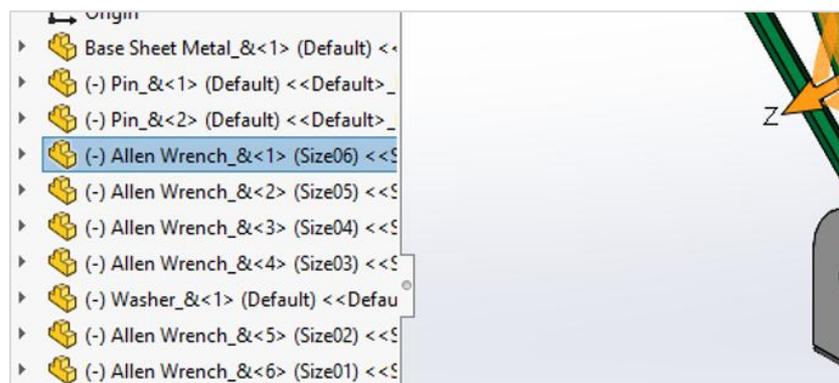
- You can similarly configure any displayed dimension (e.g., when editing a sketch, or by clicking a feature to show its dimensions), again, select the dimensions, right click and choose *Configure Dimension*.



- Making Configurations for special purposes is useful.
 - For example, make a config named FEA for a SolidWorks Simulation study. Add a split line circle to make a small surface for applying a restraint or load. Suppress this feature in other configurations and un-suppress it in the FEA configuration.
 - Make a configuration for simplified drawing views with certain features suppressed.

Assembly Configurations

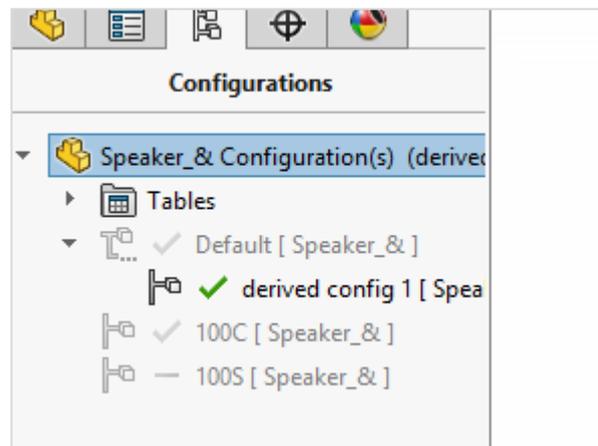
- Components (parts/subassemblies) can be configured the same way as features in part select the component(s), right click, *Configure Component*.



Derived Configurations

These are child configurations of another configuration. They inherit from and change their geometry following the parent unless a specific change has been made to a feature or dimension (say a smaller hole size).

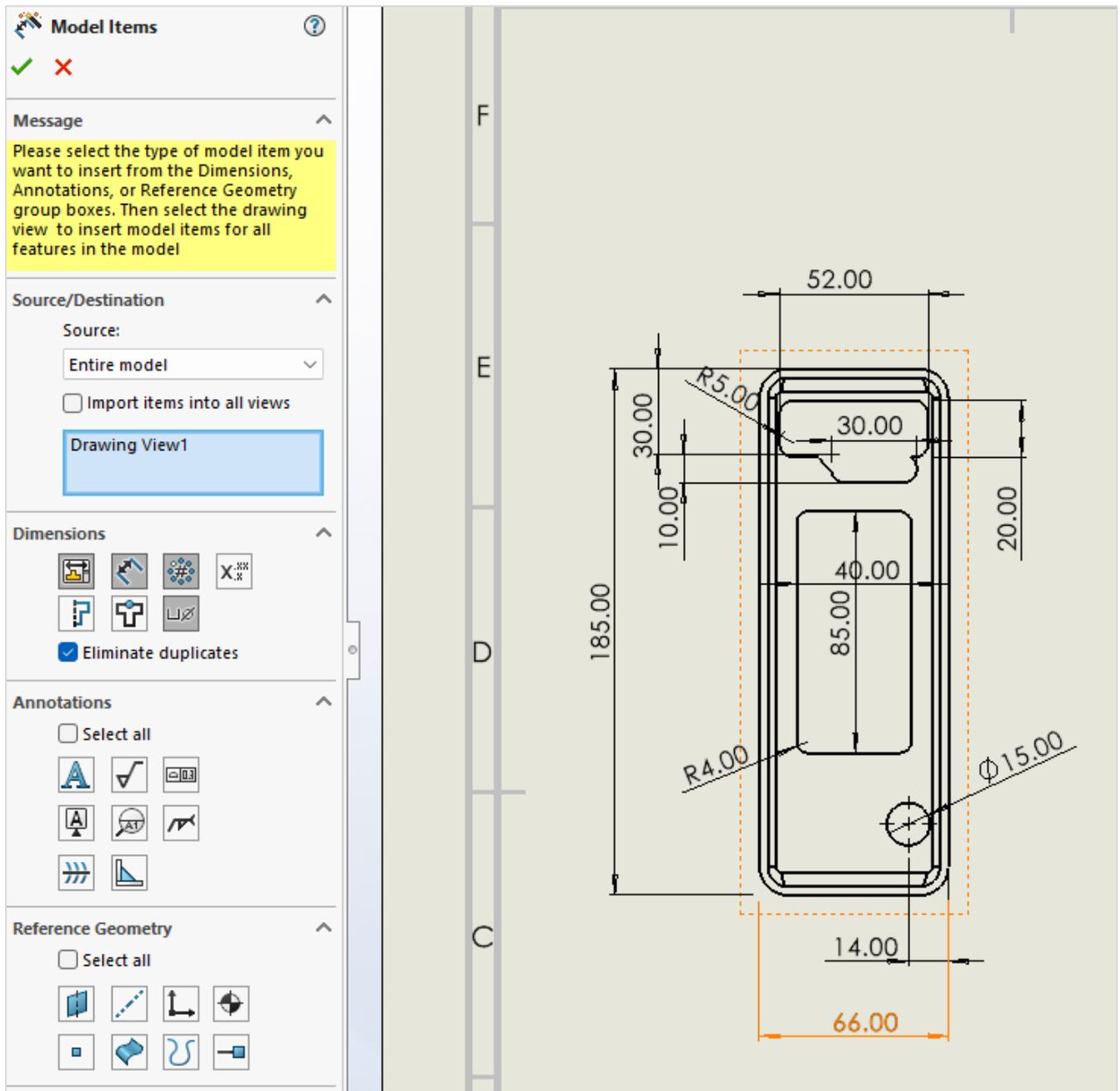
- In the configuration Manager, Derived configs are created when another config is pre-selected. to avoid creating a Derived config, make sure that no configuration is selected when you create a new one.



SOLIDWORKS Drawings

Professional drawings are essential for communicating design intent and ensuring successful manufacture of a product. The guidelines in this section outline key methods for building high-quality drawings that are easy to understand, reliable to use, and efficient to edit without compromising on performance.

1. Drawing layout, views, line-styles, and dimensioning should follow (or be based on) a known standard, e.g. BS 8888. ***This ensures consistency in drawing appearance.***
2. Views should be labelled for clarity, e.g. descriptions and scale (where required). ***This helps the reader.***
3. Import Dimensions using MODEL ITEMS where possible to reuse existing model definition dimensions rather than manually creating each one. ***This is a time saver, increases dimension robustness, and allows editing model dimensions directly from the drawing (if desired).***



4. Use dedicated layers for Dimensions and Annotations to allow for colour/weight control. ***This is a time-tested drafting practice.***
5. Switch off *System options > Drawings > Performance > “Include detail mode data when saving”*, UNLESS using detailing mode for LARGE drawings/assemblies. ***This is a performance enhancer.***

SOLIDWORKS File Property Recommendations

- Fill in model file properties so they be used in drawing borders / title-blocks.
- Fill in model file properties so they be used in BOMs.
- Fill in model file properties so they be used for Searching.
- Fill in drawing file properties where appropriate.
- Modify properties.txt file to use company properties needed for drop down in properties. ***This ensures proper spelling of property names for extraction in drawings.***
- Template files should contain common file properties used in your company. ***This is a time saver, and it ensures consistent properties in models/drawings, plus provides good data for searching in a data management system.***
- Use Property Tab Builder to make filling or selecting properties easier. (a data management system would use Data Cards instead). ***This is a time saver and makes for easier and more consistent file property entry.***
- If SW Configurations are used in models, decide what properties are configuration-specific. ***This matters when extracting properties to title block notes or BOM columns and it ensures that the correct information appears in the drawing.***

SOLIDWORKS Data/ File Recommendations

- Ensure files are given unique filenames even if saved in different folders. ***This ensures file references are unique, e.g., when an assembly is opened, the file references loaded for standard or common parts are not accidentally pointing to the wrong model.***
- SAVE often – CTRL Q (Whole model rebuild), then CTRL S. ***Saving often is a good practice for ANY software.***
- Save data to a location that is backed-up often, e.g. network server. ***You only have to lose design data one time to appreciate this.***
- Network locations are slower to load data from compared to a local disk, but that data will be safer as it should be being backed-up. ***These are the pros and cons. A data management system provides the best advantages of network and local storage.***
- Ensure the backups are made are tested regularly, i.e. you can recover files from them.
- Setup Auto-recovery and Backup files to local disk, e.g. C:\SW Recovery & C:\SW Backup.

- Use shared/common Templates, Sheet Formats, Material Libraries, etc.
- Use shared Design Libraries for common Library Features and Parts.
- TIP: SOLIDWORKS PDM should be considered for central file storage and local file editing (Check-out caches file from server, Check-in copies it back to the server; PDM server backed-up daily.)

SOLIDWORKS Graphics Recommendations

- Switch off Real View graphics and shadows when modelling to reduce eye fatigue. Real View does not aid in day-to-day model visibility, and it requires lots of computational horsepower.
- Keep graphics drivers up to date to ensure best performance out of your hardware.
- If the graphics card is supported, use *Tools > Options > System Options > Performance > Enhanced graphics performance*. ***This option improves graphical performance and affects rotate, pan, and zoom for parts and assemblies, and the display of drawings that have shaded or draft quality views.***

Conclusion

Adopting these SOLIDWORKS best practices will help ensure that models, assemblies, and drawings remain robust, efficient, and easy to maintain throughout the design process. Continued commitment to these habits will lead to better-quality designs and a smoother workflow for all users.

For additional information and guidance, please don't hesitate to use our support services.