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FDM DESIGN RULES

For Repeatable Designs of 3D Printed Parts

Abstract

The latest draft of a set of rules & steps to produce high quality engineered mechanical parts with FDM manufacturing. The example parts published by DM all follow these guidelines which minimize the decision-making in the development of new designs and generate highly-repeatable digital models with research-grade quality. Download this guide at qr.net/openlabproject and access my example parts for free.

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A Policy Guide for 3D Design

This is a design guide for my personal design policies for functional parts that are repeatable, well-engineered and carry digital hygiene as a software programmer would include in a software developer's guide. This rule set is not for every possible 3D model, but a subset of all parts that overlaps printability, parametric design, and best practices for FDM manufacturing.

The goal of revision 1 is to write something worth improving in the future. I anticipate many iterations to follow. After a decade of practice, it is now appropriate to cover the most common aspects that recur in the open-source design collection I've titled "**OpenLab Project**," and which recur in "**SCUTTLE Robot**" designs that have spread across the world.

- Design for manufacturing
- Design for FDM printing
- Design for open publication
- Integration of off-the-shelf parts
- Compliant mechanisms design
- Design of assemblies
- Functionality-driven design

Convergence of the listed topics forms a stack of goals & limitations in one space. Several barriers and several synergies occur between one goal and another goal. Some exciting outcomes of optimality arise, that satisfy multiple needs at once. From my decade of design under the listed conditions, I created routines that usually satisfy all the necessary needs to make functional parts and found consistent choices that can make repeatable parts. Now the set of rules and guidelines are becoming apparent enough to write down along with their reasons and limitations.

Definitions

Understanding a model: to gain the level of study on a model such that one can rebuild the model from scratch, and one can modify a model with full readiness of the original author.
Pinned joint: a joint not only implementing a pin but also having greater contact between pin and inner diameter than the contact between jointed faces. discussed as a subcategory, in friction joints.
Complicating a model is to add features, add dimensions, or data which could still be removed while maintaining the design intent.
Ready or Readiness is a binary criteria summarized by the quality and completeness of this part is sufficient to post. If the part is ready, then the open source collaborators have power to A) gain the utility and B) improve the model.
Worry: to perform engineering effort aimed to countermeasure and undefined or unvalidated problem.
Post: A publicly accessible repository of a model of a design, such as a GrabCAD upload. The post can be differentiated by its hyperlink, or by its' author and post title.
Overbuild: the excess material in FDM parts beyond the design geometry which exceeds the nominal size of a part but is not to be corrected by adjusting the design.
Postprocessing: manual work on a part after the digital fabrication stage (ie sanding a 3D print).
Off-the-print (OTP): The part which has no postprocessing, immediately after FDM printing.
Feature: a nameable element inside of a physical part that is also an element in the feature tree of a CAD model. An answer to a function.
Function: a behavior sought in the design such as "hold a screw" which requires a feature. The child of purpose.
Purpose: the parent to one or more "features."
Part: the nameable object which also maps to a CAD model.
ODOP: one design one part. Specifying a level of simplicity where the design Rules herein are relevant.

Overarching Goal

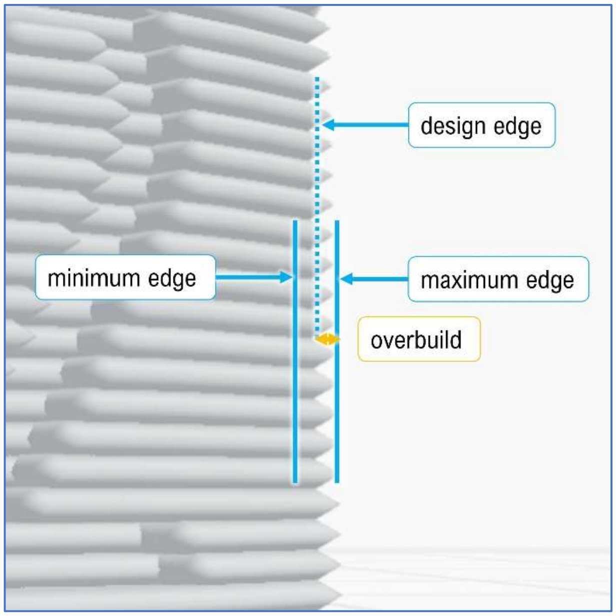
Our goal is to fully define all designs that we produce as mechanical engineers. This is a precursor to all science and engineering progress. Everything that we call "approved" is fully defined; that is the goal. You might design a thing with groundbreaking performance, but if the design is not fully defined then nothing else matters. We can only improve a joint if we know the adhesive. We can only improve a diagram if we know the designed layout. We can only write a torque spec if we know the fastener material. We can only integrate a genius subassembly if its dimensions are deterministic.

Therefore, the rule set is a reference which populates all the design questions which are possibly missing from the design documents. If an aspect of the design is in question, the answer lives here instead of in an assumption. Any "Openlab" tagged, ready design is defined fully between this document and the published documentation. As of 2025 most designs fall short of meeting this goal and the long-term aim is to reach it.

General Tolerances:

These are the assumed characteristics in printed parts that are "off-the-print"

Overbuild is anticipated to be 0.3mm on each face and 0.6mm for a part, as a general rule at the conservative level.

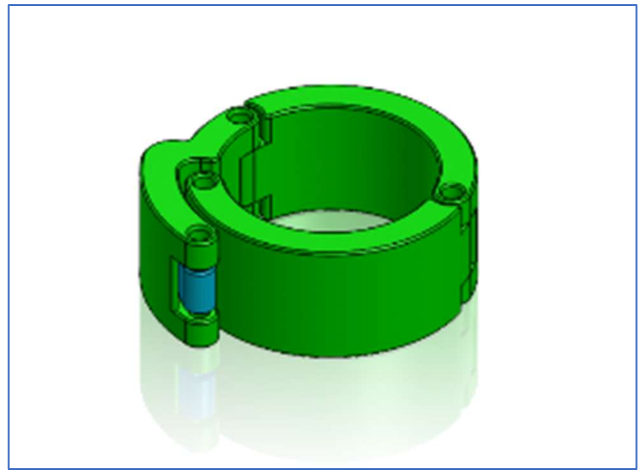


Examples

For the 2025 period, most breadth of topics will be simply a list of examples with accompanying guidelines. There are ways to categorize these examples but topics will overlap heavily. This year I finally gained enough experience that my new designs are made up of features which map to a consistent set of rules. As of 2025 September I have not recorded rules so this list will help populate them.

Compression of parts

The following compression part has deviations that can stack up so it is better to separate the compression target from the design geometry.

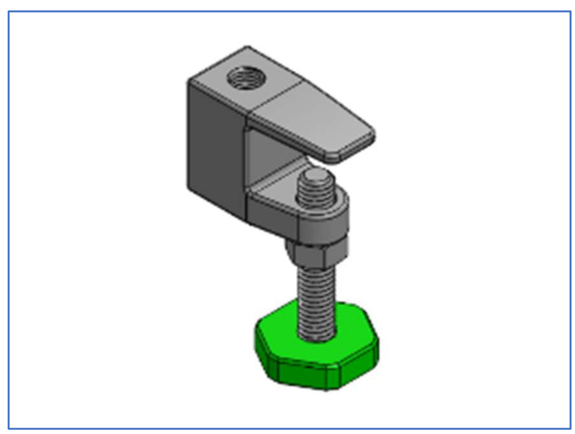


For this part called "collar" the diameter in the design would simply be made as a clearance size around the tube to be compressed. Then, a separate component such as a thin film of rubber or cork sheet would be applied.

We can deboss a region to consume part of the thickness of the cork, so 6mm cork pad may sit in a 5mm deboss and we target 1mm of compression.

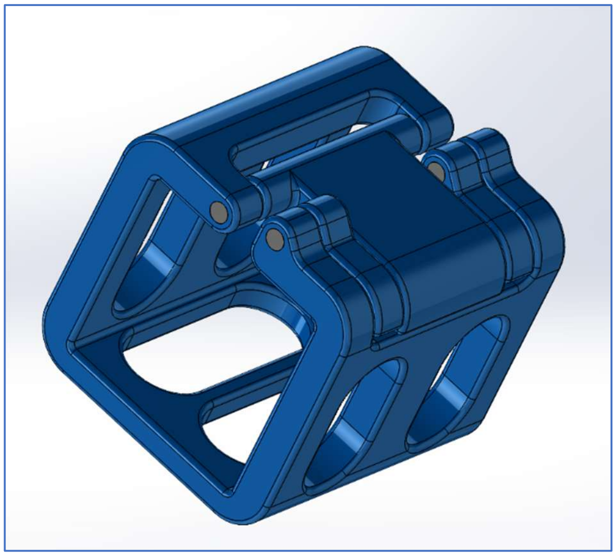
Constrain before Bonding

When combining two materials, it is best to fully constrain the two parts before bonding. This example shows a knob item added to the head of a machine screw. The (need picture) mating surfaces constrain the green part WRT the bolt in all directions before we add an adhesive. This means the adhesive is not to battle forces, only to hold the parts from slipping.



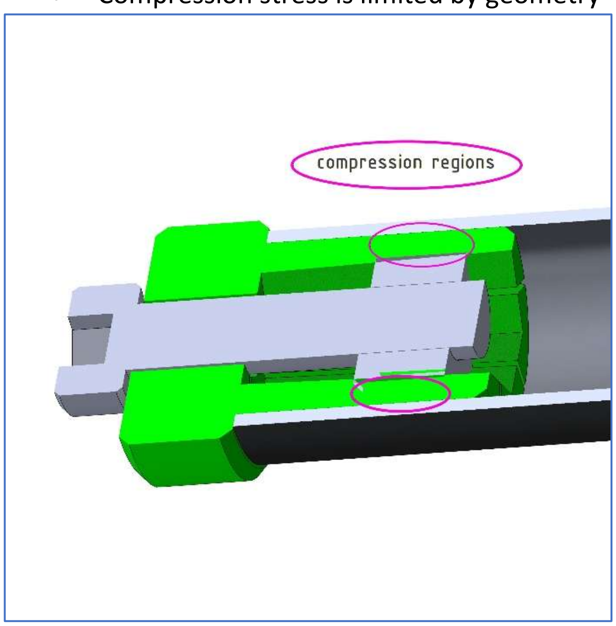
Compliance as a Linkage

For achieving the behavior of a 4-bar linkage, compliance can be used as a linkage. This is an example of merging a four-bar-linkage design and compliant design. The linkage function is only to move the assembly a small distance so then one solid member can tolerate the movement of opening and closing and remain in the elastic region of deformation.



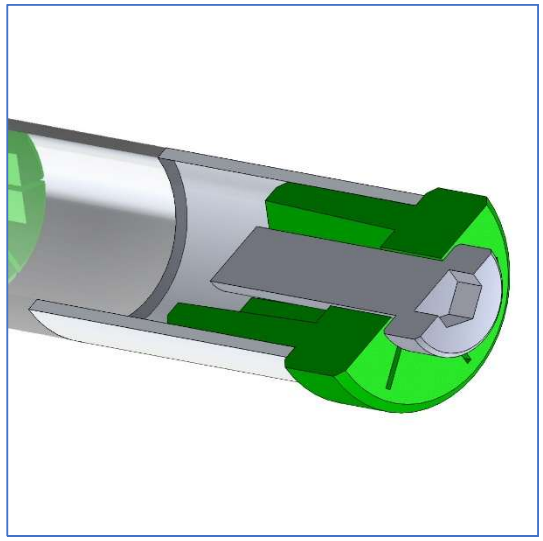
Compression Loads
If a printed part will be loaded with high pressure, this should be compressive force only. Mandrel.sldprt, shown below, performs a job of fixing a tube in place when assembled. The job is achieved by friction, and friction is achieved by compressive load on the plastic. That load takes place between the steel nut and the steel tube wall (interior face), with compressed regions circled.

These rules are implemented:
- High stresses on are compression-only
- Compression stress is limited by geometry

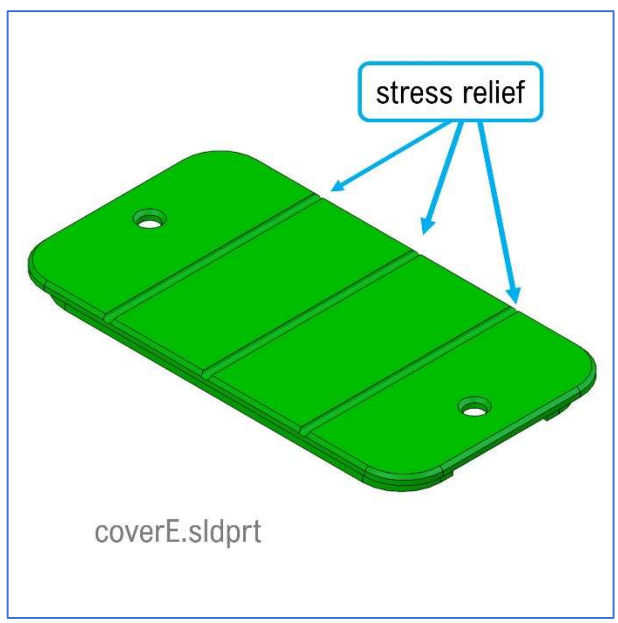


This part titled mandrel creates a high stress on the ABS plastic but only in compression. The squeezing of the internal nut (not shown) against the plastic pushes it against the steel wall. It is a general rule not to make a high stress state of the polymer a direct function of the part, but in this case we know we are only compressing the plastic.

We limit the maximum possible compressive stress due to the difference between the tube inner diameter and the thickness of the plastic at the thickest region of the taper.



Curling Relief
Flat planes resting on the printer bed are under stress due to the steep temperature gradient in the vertical direction. The bed is hot (below), and the next layers are cool (above). This stress can be relieved by segmenting the part between areas of the base layer.



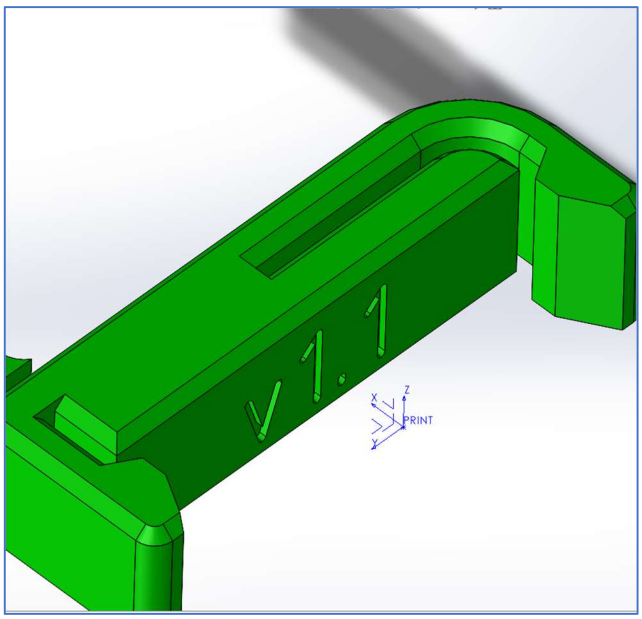
The standard feature for stress relief at the base is a semicircle with 2mm diameter, and creates an overhang that requires no supports.

Print Coordinates

Each part model begins with an origin and a coordinate frame by default. We add a coordinate frame called "PRINT." The PRINT frame is defined in terms of the native coordinate frame, by the user. The global frame is fixed. During Modeling, the TOP plane points to the sky, and the model is oriented as it is found in a designated assembly. There are two steps to setting the PRINT frame:

1) The z-vector is set perpendicular to the TOP plane.
2) The z-vector maybe redefined if called for. The purpose is to align UP in the FDM printer. After some modeling has begun, the designer decides for certain which way the print will orient. For highly symmetric parts, this may be decided later in the model during the final chamfers & fillets.

So, for most models the z-vector of PRINT frame remains aligned with TOP plane. For models that are easier to design in another way, the PRINT frame becomes unique from the native coordinate frame and the PRINT frame is used to export the STL file.



Version Numbers

(version numbers have several layers of meaning and rules which are broken frequently so the ideal scenario is listed here).

The version number answers "how many changes have been made which actually drove a print-fit-use cycle?" This means if the part is adjusted in the model and not printed, the version has not changed. If the part was printed, and immediately failed to fit in the assembly, the part has not been used yet and remains at version 1. If the part is used, then received an improvement with no change in function, and reprinted, we have a version 2.

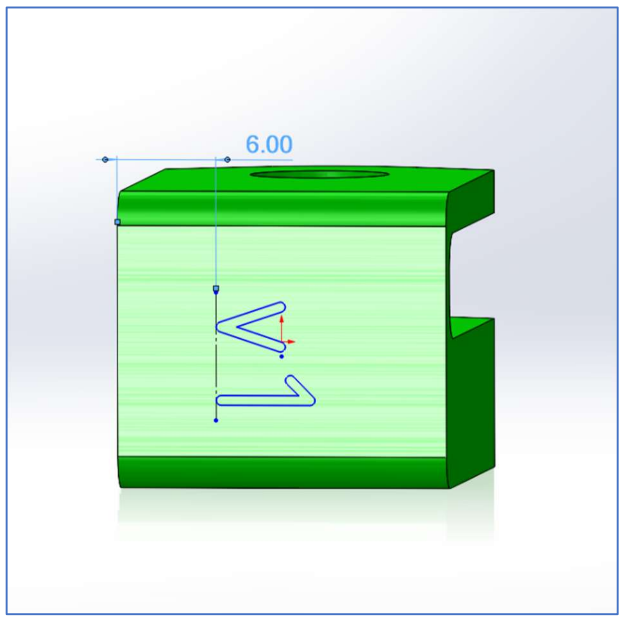
Version numbers are most important when distributing data. If the part is sent by email, posted online, or in some manner "receiving a timestamp in real life" then it ought to also have a unique version number, and that unique version number ought to reflect what is found elsewhere online with a matching version.

Version numbers are added to the part with a deboss and oriented in a way that communicates the "upright" printing direction. Debossed versions contain:

- 6mm height of text
- DS ISO Bold font
- 6mm height from base of the part
- 0.5mm depth of deboss

Version Location

The text in SolidWorks resides on a construction line, on a location that will be visible when assembled, ideally on a flat location of the part where additional space is available for text.



Design Orientation:

(to be discussed)

Print Settings and Design

Print settings must not feedback information back to the design desk! The printing process should be made repeatable, defined, and then our designs either DO or DO NOT succeed with this process. All printable designs carry intention of "printing on FDM with ABS" and if a part has failures then the failure should trace to "failed print" or "failed design." We must be able to refine our designs and refine our manufacturing separately.

Standard Settings:

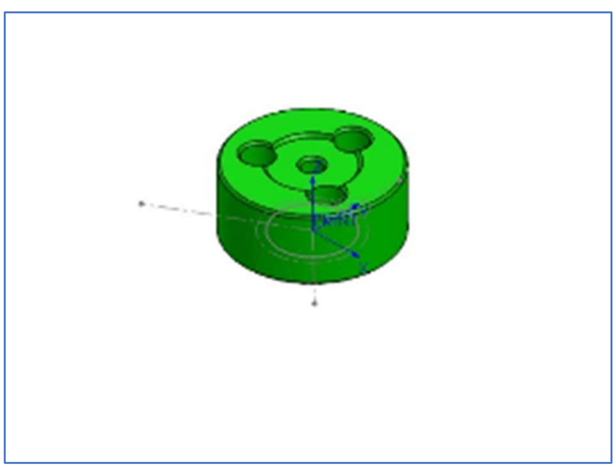
- Filament: ABS
- Infill: 30%
- Exterior Solid Layers: 2
- Path Width: 0.45mm
- Layer Height: 0.3mm

These settings are the out-of-box printer settings for ABS on Flashforge Creator Pro as well as most printers between 2000 & 2020. The purpose of this selection is A) to design along broadly-accepted manufacturing methods and B) to wrap the material and manufacturing method into a “design material” so the design-geometry + manufacturing method = repeatable performance.

Parts for Limited Lifespan

The part called “holeGuide” functions in a similar way to a drilling guide commonly found at hardware stores. This part constrains a drill bit to perpendicularity with a wooden part. In a normal design rule set we cannot allow the plastic holes to rub tightly against a drill bit due to wear. The walls will gradually lose their dimensions causing loss of function. However this part is made for a use case only a fraction of typical use-count. It functions in the assembly stage of a specific assembly, so 10 uses will not cause failure.

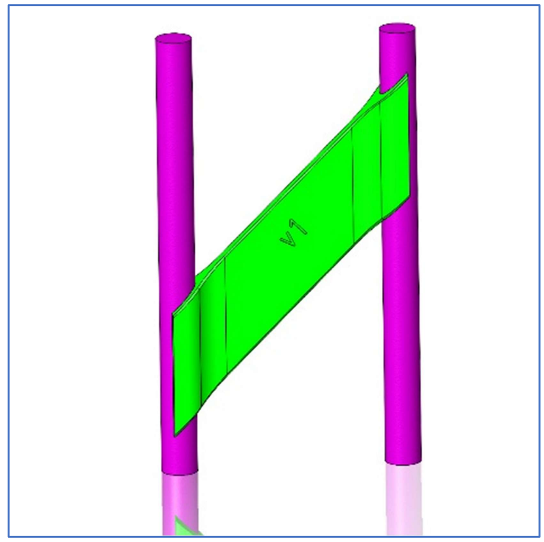
If we wish for the same part to have a permanent life then the hole diameters can be enlarged and fit with a brass or steel collar. Our global locations have high accuracy to satisfy the purpose of aligning a drill within 0.2mm global locations.



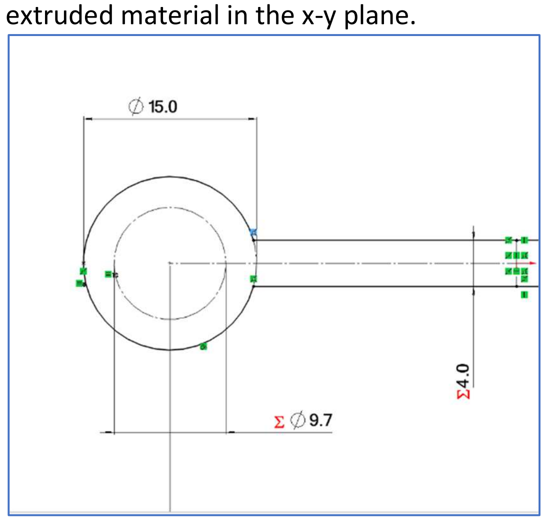
Snap-on Round feature

Perform compliant snap-in using 70% of the diameter. The “web” part makes a snap-fit with two rods and re-uses a geometry for ABS plastic grasping the round stock successfully. The general rule is to take the hole diameter and use 70% of that diameter as the arms’ reaching dimension.

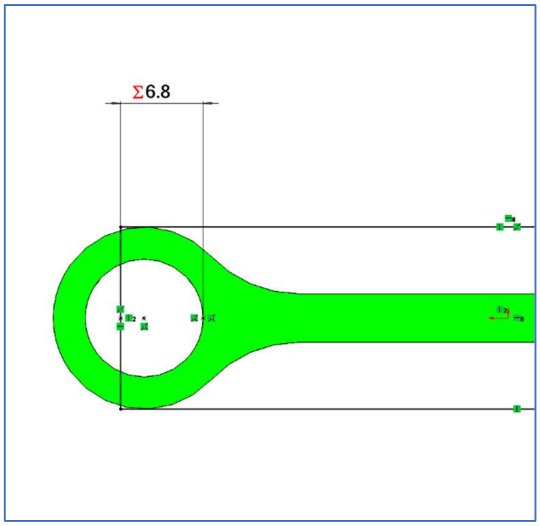
Once seated, the pole will have friction in the grip because diameters were matched (arms and pole) without any design gap. The print is estimated to require 0.6mm around a diameter for guaranteed slipping so we have eliminated 0.3mm at each side of the radius by matching diameters.



The sketch (next image) shows a designed hole diameter of 9.7mm which is the measured diameter of the rod with ordinary calipers.
The assumptions about fitness of a grip is based on the extruded material in the x-y plane.

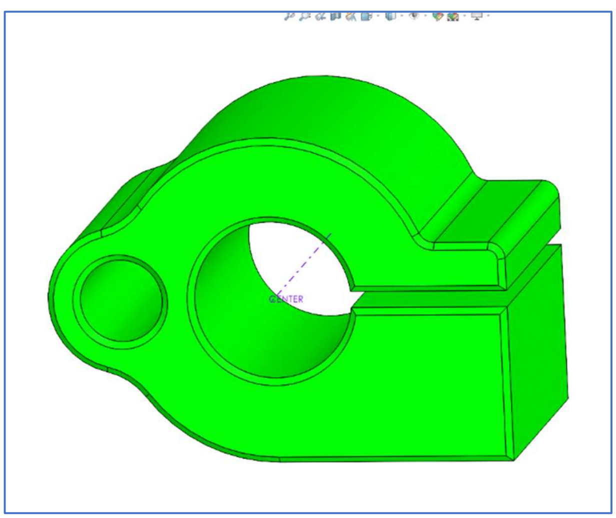


First the model is built such that the whole circumference is contacted, then the material is trimmed from the end. The cut-extrude is made with the following sketch which shows a 6.8mm calculated distance where an equation (0.7*diameter) is entered for the dimension shown.



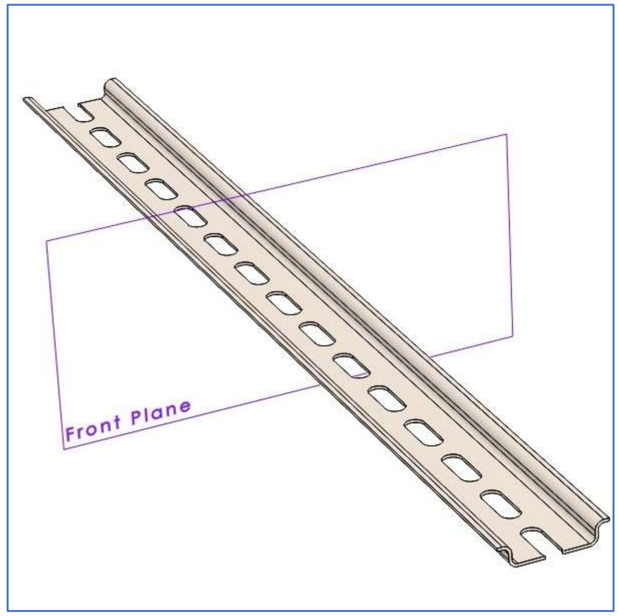
Gap, Pre-Clamping

For clamping parts with fasteners, match the perimeters to be contacted and remove about 2mm of material. That is, design the geometry in the relaxed position and install it with a clamped position. This part, titled “clamp” is to be clamped on a round tube to carry load in multiple directions. The fastener (not shown) will tighten the ABS plastic around the large tube and then cause the gap to tighten. For this type of clamping, I form the clamp interior to match the tube exterior diameter. Then I form a gap at least 2mm in width. greater than 1mm to begin with. On the example part, we get firm friction when the gap closes by 1mm and very strong friction closing about 2mm. The tube has a 30mm diameter.



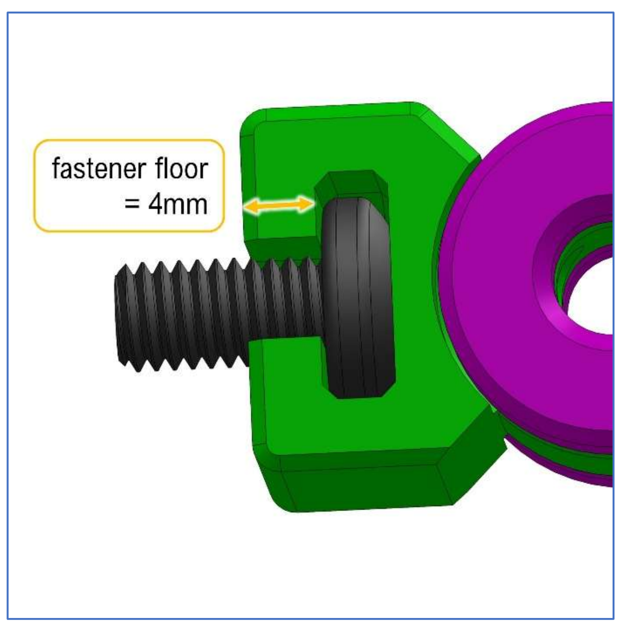
Center Datum

Place the origin in the midsection for long beam-style parts. Long stock CAD models will appear in various lengths in an assembly or in future designs, the cross-section is built on the RIGHT plane and the length is extruded from the midsection (in both directions) by selecting “extrude ► midplane” Generally, this gives a more flexible model when the parts are parametric and several different assemblies will integrate the part. There are benefits of extruding in one-direction but this choice has shown the most benefits for the most purposes later in the workflow.



Fastener-floor

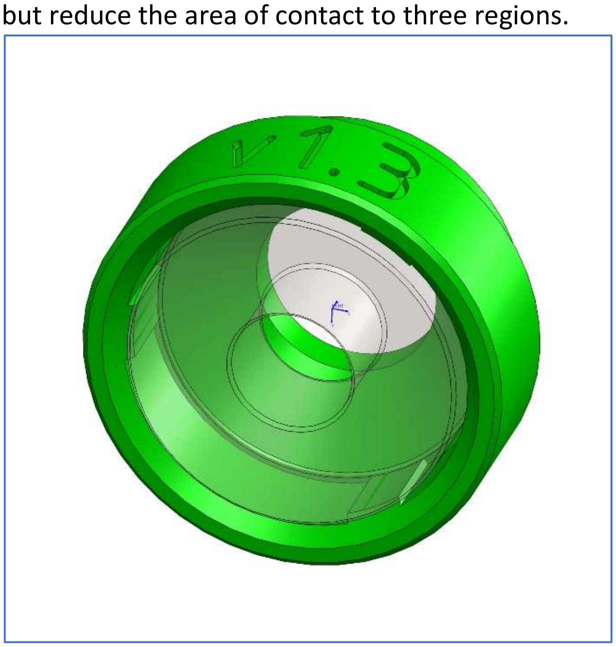
The material thickness clamped by a fastener ranges 4mm to 6mm by default. The floor where a fastener clamps down is set to 4mm to 6mm. This gives compatibility with our common screw lengths for M6 and M5, usually 10 or 12 mm. The floor is the region that rests below the fastener head and becomes compressed when the screw is tightened. The model hingeEnd20 attaches to a 20mm aluminum extrusion, and the floor is 4mm.



Press-Fit Round Parts

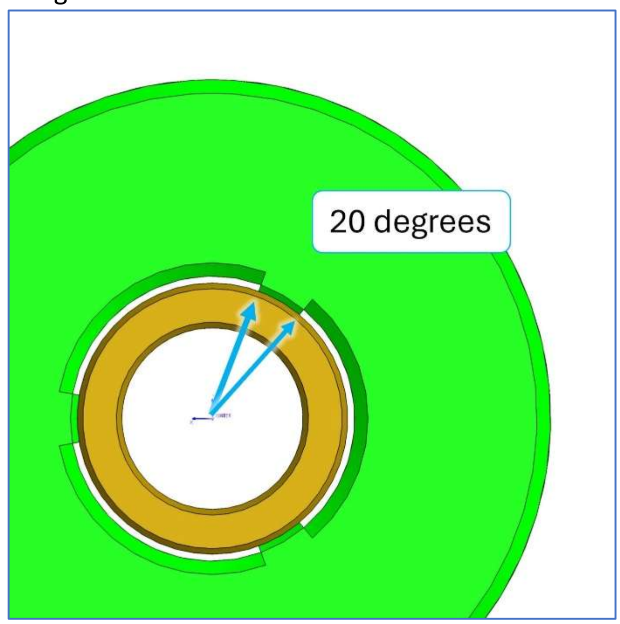
Fit two cylinders by a two-step design, based on nominal diameter D. Form a hole with diameter D and then cut-revolve a boss for D2. Choose the nominal diameter D equal to the mating part's true outer diameter, as in the example "SleeveBR" shown with a bearing in the image. See the bearing displayed with transparency.

By designing with Diameter D, typical printers will overprint form a reduced diameter that yields interference between the parts. Keep this interference but reduce the area of contact to three regions.



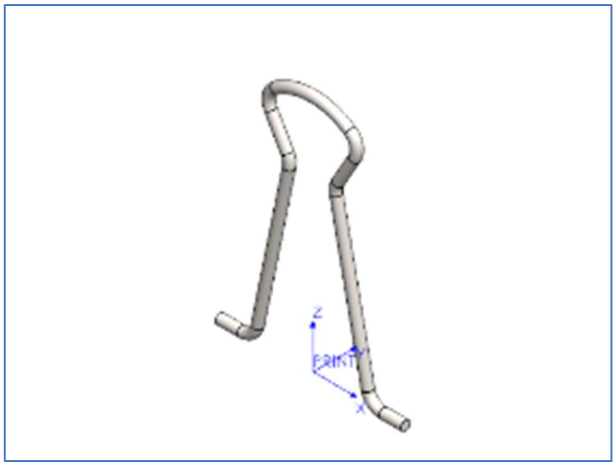
For the bearing sleeve example, we cut-revolve 100 degrees and copy this for three total gaps, and three contact patches of 20 degrees each. See the image above for "SleeveBR" which has three contact patches for the bearing. The bearing is installed in the sleeve by pressing the green part down onto the bearing.

For this design method, postprocessing is likely and we aim to minimize it. It is easier to remove material from the three small regions compared with the entire circumference. The gap regions should be 0.3mm offset from diameter D at minimum, per the general clearance-fit rules. The following image highlights the 20 degree size of contact patches used as a starting point for these designs.



Springs

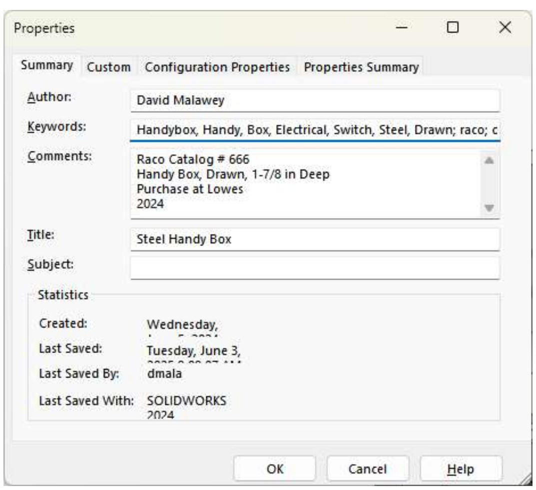
Spring-designated parts are always drawn in their natural form and the model is not enhanced with simulations of deflection etc. This spares the work effort until the part is put into use. In some cases the working part is measured at key locations in a deflected state, and then a simple line or point added to the sketch indicating where deflection is desired to end. The part shown is called spring.sldprt



Standard Origins

A part which has its full geometry rooted in a public standard and which has it's dimensions rooted in real life-parts – this type of part is uniquely valuable for modeling and choosing a master configuration. For this type of model, we break all skill-based limitations in the modeling of this part because contributors have access to the true geometry. If I post a design which has shortcomings, those shortcomings can be corrected by a high volume of community designers, hugely boosting the chance of improvement. Therefore, the existence of the post becomes more important than the correctness of the model.

Handybox.sldprt is one of these parts which deserves high effort in modeling and planning. The



Circuit Board (PCB) Abstraction

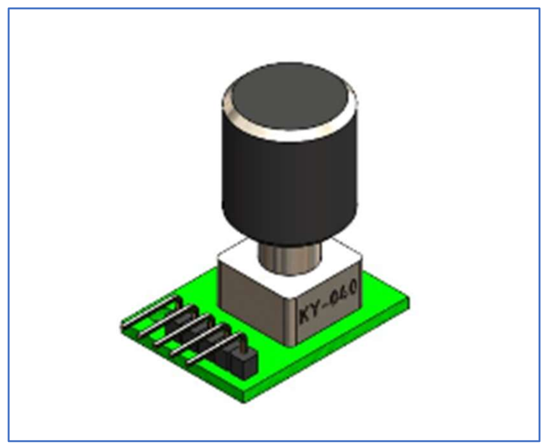
When designing a bracket, the component to fit on the bracket should have a simplified model. The circuit below features a rotary knob and like many boards it's nothing but a host for one key sensor and the terminals to connect it. The goal is to have a minimal model which visually identifies the part, and includes the key information:

- minimal features
- exterior board dimensions
- mounting hole locations
- the tallest feature (for clearance)
- the interface locations, like protruding ports

The interface locations are places where assembly-level design must offer clearance, or mate, or access for a tool. The features at interface locations are to be modeled.

The identifying features of the physical part are those which can be seen in a thumbnail image and are not required for functionality but are required for a "ready model."

A board made of an assembly which can be changed may have geometry of interest.



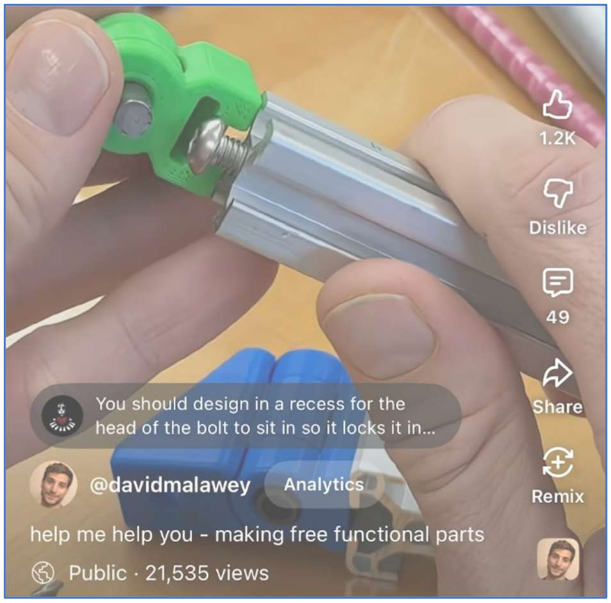
Worrying Not Permitted

Worrying: implementing a countermeasure for a failure mode which lacks validation through testing.

While designing a part, we should not add material thickening or geometry that makes the part "extra strong" on the first trial. If there is a concern for strength of a feature then produce the geometry that serves the function and then gather data about the limit of strength or other failure concern.

Worrying produces artifacts in a design which obscure the design intent & often carryover into revisions wherein the design focus is on another feature, so the unnecessary solution does not later become omitted from the design. It remains captive, potentially forever stacking up the added cost & material across iterations & space & time.

In the following image we have a design suggestion from a peer to add a feature to this joint part. The suggested feature is in the worry category if the "locking of the bolt head" is the solution while "slipping of the bolt head" is not a verified issue.



The impact on this particular part: This part has fairly complex model geometry such that it is challenging to understand. The average user is likely to copy the design without studying and understanding the model, which is no problem. In this case, the user is also not prepared to omit features, even unnecessary ones. Our goal is to publish the most simplified variation of the model. Then, users can worry more complexity into the model at their desk even though they can't worry complexity out of the model.

Orphaned Functions

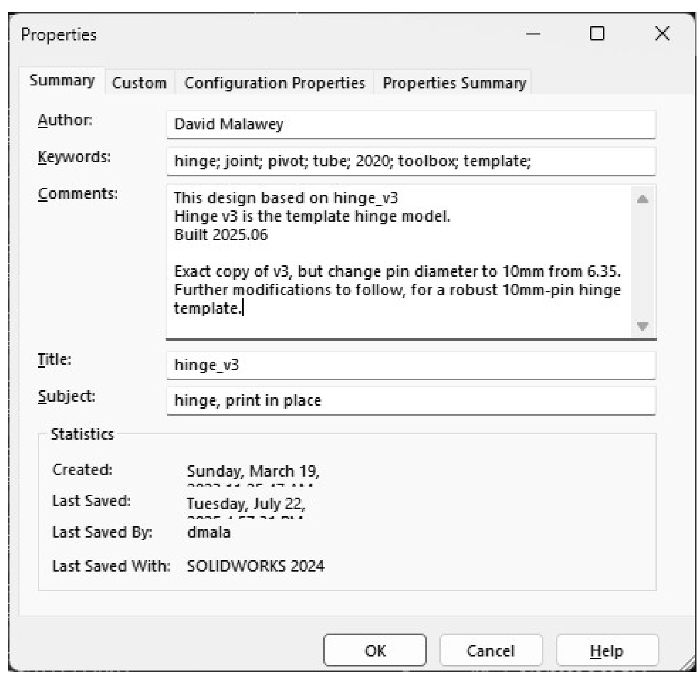
The joint part in the above photo has two functions, those are [fasten to the extrusion end] and [perform pivoting in one DOF]. The functions may individually appear in other parts. Firstly, the design should be simplified to execute one function instead of two. But such a change would break the total function of this application. In this case, each of those two functions should appear in a "representative part" that is designed separately, and features the highest level of refinement and simplification that can be made.

If we publish a part which carries a function but this function is in need of a representative part which isolates that function. Without a representative part, we have created an **orphaned function**. The representative part is the only way to direct the community designers towards improvements and allow re-integration in their own separate designs.

The following image depicts a reduced model of a hinge which retains the refined singular function so it is simply called Hinge V4.

For a representative part it becomes crucial to note that it is a "master model." This note should be included in the same directory as the model, or better yet, in the same document. For this Hinge_v4 I included information in the "comments" section shown below.

Without the comments, these representative parts multiply across the hard drive in many locations having matching keywords and can cause the purest model copy to become lost.



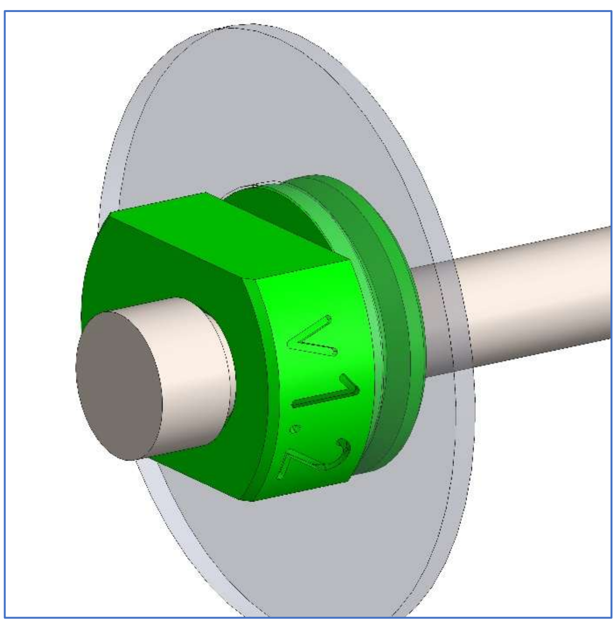
Informing Bodies

Many designs have a core function fully dependent on a mating part, and the mating part has a simple geometry. In this case we add the mating OTS part into the solidworks design, such as the example “sleeve.sldprt” which must mate with a thin wall and a bolt. This part is drawn in the standard green color and the two mating parts have been included in the model.

For these models, the informing part is omitted from the assembly.

How do we decide to include or omit such geometry? Consider three factors: first, the mating body can be made in one or two features (low effort). Second, we think of representation enhancement: the design has low chance of causing confusion when released, because this belongs in a well-documented assembly. Those who access the model are likely to find the other components and images that show the full assembly. Otherwise, they may understand the bolt to be part of the model. This geometry is more likely to help than harm the representation. Next, the users are likely to make geometry adjustments to this part, so the bolt and thin wall carry the original mating-part dimensions, rather than losing that information with changes.

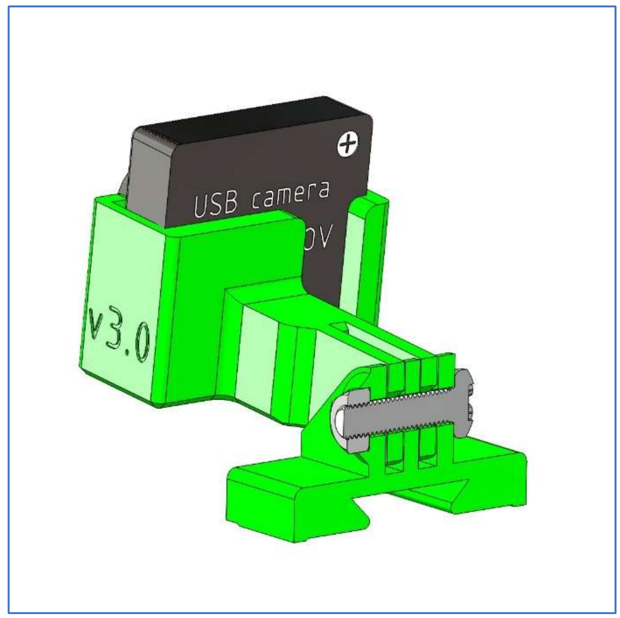
Thirdly, the informing features should not cause downstream design issues. They can be formed without any dependencies on the feature tree, and omitted at the end of the feature tree with a “delete bodies” function. Then, users can delete these bodies easily or the designer can delete them at any time.



Friction Joints

A friction joint is a joint which has a freedom of movement with a desired minimum friction in such movement. **Fingers** in a friction joint are the two-dimensional protrusions in a cross-section of this joint which interlock with a pin.

For adding friction without adding clamping force, the number of fingers is increased. The friction increases by a multiple equal to the interface count, given the same clamping force. If the finger faces are to control the friction level of the joint, then a clamping force is a control mechanism, as shown in the camera pivot below.



Friction Joints, Pinned

A friction joint which carries more friction against a pin is called a pinned friction joint.

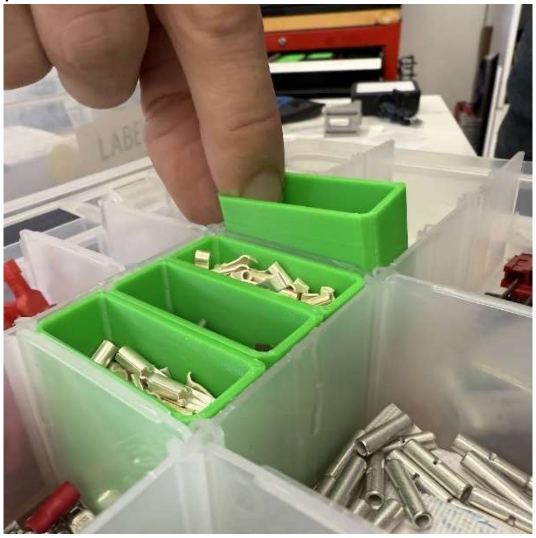
If the pin of the joint has threads, then the hole size is selected as **clearance** and the pin size is selected as nominal major diameter of the screw. For example, the M5 screw has 4.7mm major diameter and a 5.0mm nominal diameter and the clearance condition calls for 0.3mm diametric clearance so the designed hole size is 5.3mm.

Thin Wall Thickness

Thin walls are walls which serve their function by existing prior to any requirements for load. These are selected at 1.6mm minimum (this accommodates a debossed label at 0.5mm)

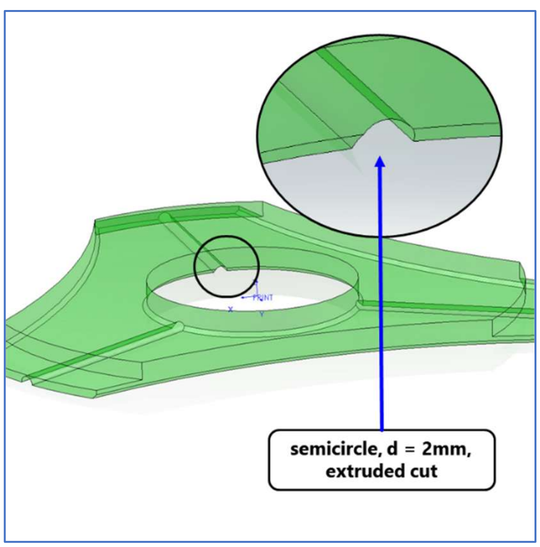
Wall thickness is minimized at the start of the design, or increased to 2mm if rigidity-requirement is of importance.

In the “metaBin” shown in the photo, walls of 1.6mm give the greatest volume, fastest print time, and ability to extrude exactly 4 paths at 0.4mm nozzle width, in FDM printers.



Flat Areas

Flat areas contacting the print bed pose a risk of curling. The areas are segmented to relieve stress in the layers near the base of the part. The part below is a flat plate which mates to a 6-inch PVC pipe. Use a circle profile to extrude a cut to segment the parts. Usually this gives some positive aesthetic to the geometry and applied in a symmetric way, making smaller contact patches with the build plate that have equal area. This is applied for areas larger than 40x40mm.

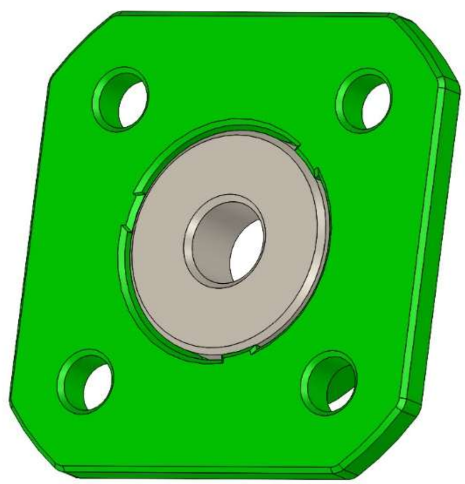


Retention with concentricity

Retaining a round feature like a bearing is made with three contact points. Clearance size is chosen for the noncontact region and interference size is chosen for the contact regions. The contact regions are made rotationally symmetric with a parent feature designated “vertical” and aligned with a main plane such as “front plane.”

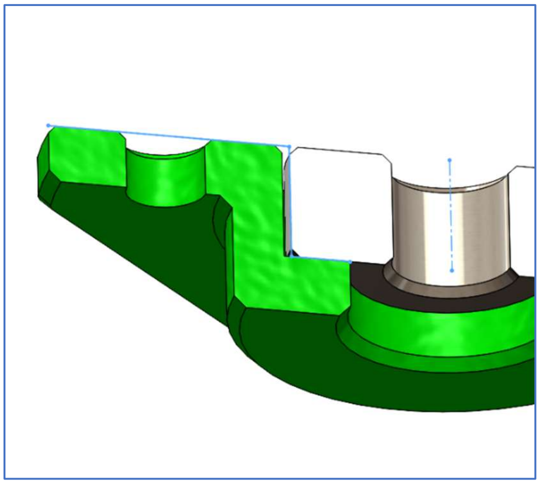
The Hub shown below performs retention of the bearing and achieves concentricity when the mate is realized.

The disc retention is evident in the next photo: see three revolved bosses which contact the bearing.



In this design, the bearing diameter drives the design size so this diameter is drawn into the model at the earliest stage of modeling. Sketches are higher priority than feature parameters so the sketch defines this geometry.

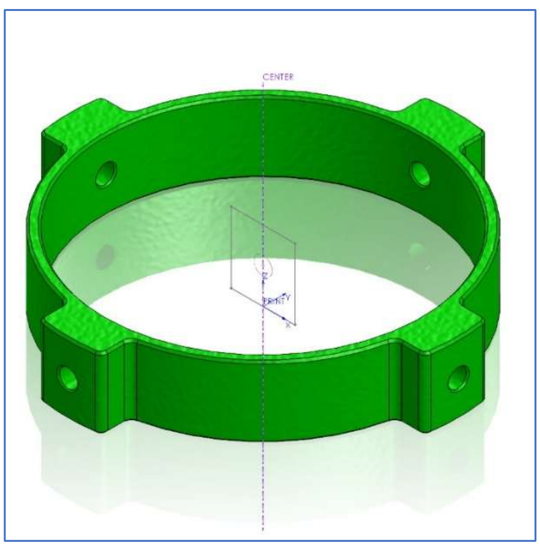
See the blue line in the next image which is revolved and thickened to build the solid.



(above) see the sketched line in blue, which is the minimal sketch to define the body geometry. That is, followed by a revolved "thin feature" with a thickness attributed.

Holes in round tube - Fabricating

If holes are to be designed in round tube then they are designed symmetric at 180 degrees. This alleviates effort in any drilling & tapping operations such that the technician can drill holes which pass through the part. Measuring locations at 180 degrees is far easier than other spacing selections. See the part called "holeGuide" which fits onto a PVC tube and creates a drill guide at 4 locations.



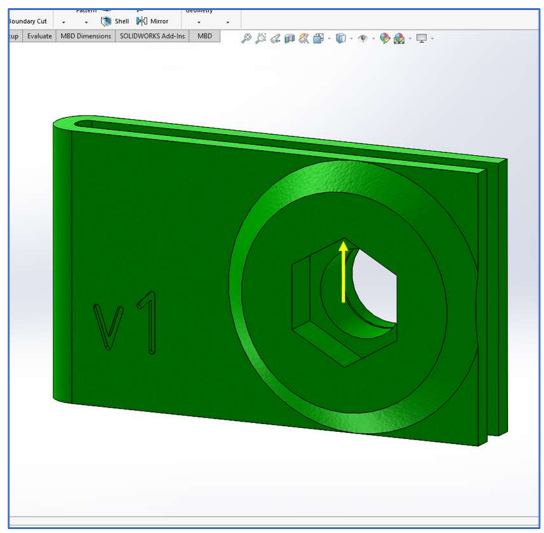
Compliance: Sigma Compliance

Sigma compliance is an arrangement for a compliant part to secure another part in 3 degrees of freedom. The part is mirror-symmetric, where the left/right sides include compliance in the weak direction, and a spring force is designed into the part in a "strong" direction.

The "gripping" like a human thumb + finger has a light force. The "clamping" like a spring action draws the part to a backplate.

Fasteners: Upright Pockets

When the hexagon pocket is produced for a captive nut it is 0.3mm oversized from the nut size, and it creates an overhanging region in a print. The angle of this overhang does not meet the 45 degree rule, so the rule is to align the shape vertically from center to peak as shown in the image of ReducerB.sldprt.



Part Appearance - Texture

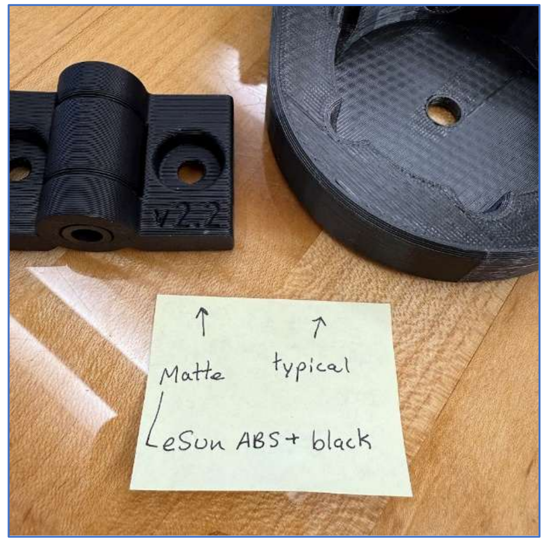
The appearance selected for the part aims to reflect the texture of the 3D print. Textured appearance is optional but it becomes more important in certain situations. During modeling, this cues the designer into the grain direction of material. In publication, it communicates about the intention of the part. In the image below, the layer lines are produced in SolidWorks by adding a texture called "PW-MT11150" which is then stretched thin and wide (aspect ratio 6x60mm) and applied with the "projection" to show layers aligned with the printing plane.

FDM Filament Color

The color of filament has an impact. Green is the selected color for demonstrating parts as it stands out from the other common parts and it reflects more light than the other colors. If the parts are subject to low light, then the green parts still show detail. In 2020 I examined the photo-compatibility of filament colors and I found green to perform best. The criteria was to check readability of the debossed labels on parts and our green parts perform best. Most maker environments have low lighting compared with photography needs, and home photography goes further with this selected green ABS, which is GizmoDorks brand. The brand CCTREE has a near-match with their Green ABS Filament, 1.75mm.

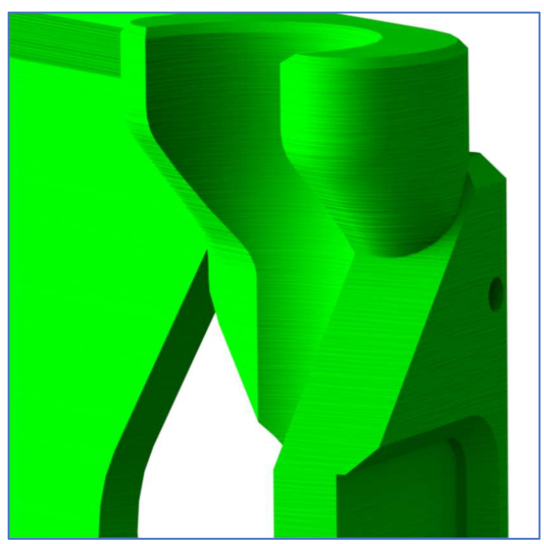
If the part requires a black color filament, most black ABS has a high gloss. In good lighting, this color highlights all of the artifacts of extrusion which is undesirable. To instead highlight the features of a design, we benefit with a matte-black material. The eSun ABS+ black filament is an example material with this property.

The next photo compares a matte black part made from ABS+ and glossy black part which is made from an estimated 9/10 of any black ABS selections. The matte black however does not perform precisely equal in adhesives compatibility due to the ABS+ formulation and further study is required to form clear answers about the chemical differences.



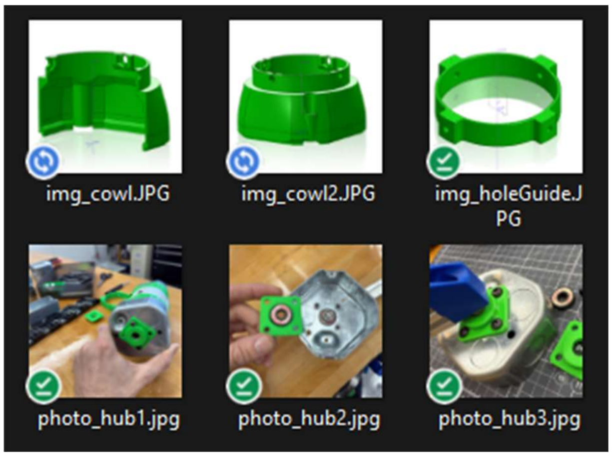
Each part is made from a template .SLDPRT part file that has a custom material applied. The material SCTL_ABS has a texture and the green color. This material appears in renderings (or with SOLIDWORKS Realview activated) with the lateral thin contours that represent the layer lines of a manufactured FDM part. Achieve this texture by starting with WHITE_PW_MT_11050 and changing the aspect ratio to 6x60 mm for the bump mapping.

Below, find an image of a rendered part with the textured material. When the design is documented, this texture appearance is a strong clue for the print direction of the part. The print direction is crucial for part strength and the visual display of this direction makes a major improvement for success of new adopters of the part, or for those who will print copies for their own projects.



Part Images

Images are made in the square aspect ratio with a resolution of 2048 pixels wide. This offers the maximum flexibility in documentation. See the images in a directory in the screen snip for an example. Photography can produce other popular aspect ratios but most screen-snipping softwares can constrain images to square out-of-box and powerpoint slides are easiest to control with square images. Inside the Solidworks menu, the default .jpg export will have a lower resolution but it can be increased above 2048 and adjusted to square A/R for a rapid imaging process. Select "File ► save as ► .jpg" to take the immediate solidworks screen and save it in a nice resolution in the part documentation.

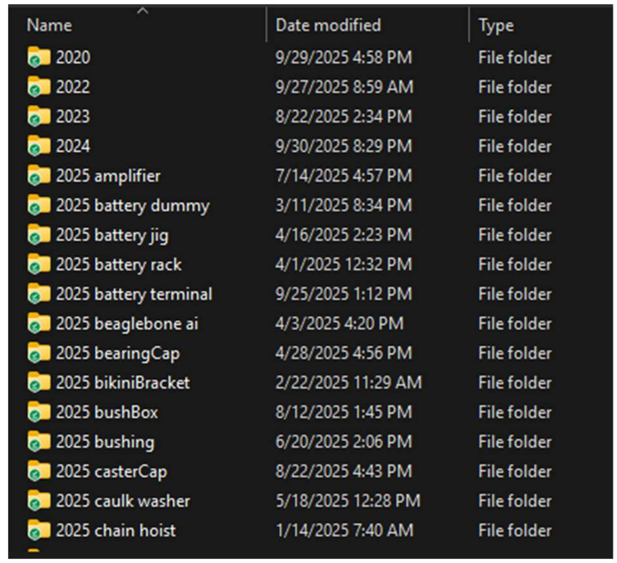


Parts Trees

Initiating Parts

Each design attempt gets a folder with a unique name and the year of the work. The name is descriptive of the part and the goal of the part. If the design goals are met, this folder is sorted and uploaded as a post. If the design goals are unmet, the folder resides with information about my attempt, and source documents. Source documents include a Solidworks model, and from that model I've generated STEP files, STL files, or other files. The Solidworks file is my design file so it should contain the full information used to produce the other formats.

It's impossible to recall the names of all parts but given the year and dated files it's easy to find an old design. After a few months of designs in 2025, the folders from 2024 are consolidated into a directory called "2024."

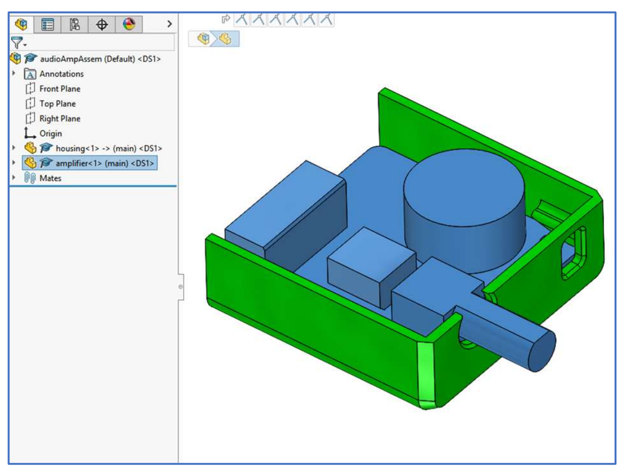


Name	Date modified	Type
2020	9/29/2025 4:58 PM	File folder
2022	9/27/2025 8:59 AM	File folder
2023	8/22/2025 2:34 PM	File folder
2024	9/30/2025 8:29 PM	File folder
2025 amplifier	7/14/2025 4:57 PM	File folder
2025 battery dummy	3/11/2025 8:34 PM	File folder
2025 battery jig	4/16/2025 2:23 PM	File folder
2025 battery rack	4/1/2025 12:32 PM	File folder
2025 battery terminal	9/25/2025 1:12 PM	File folder
2025 beaglebone ai	4/3/2025 4:20 PM	File folder
2025 bearingCap	4/28/2025 4:56 PM	File folder
2025 bikiniBracket	2/22/2025 11:29 AM	File folder
2025 bushBox	8/12/2025 1:45 PM	File folder
2025 bushing	6/20/2025 2:06 PM	File folder
2025 casterCap	8/22/2025 4:43 PM	File folder
2025 caulk washer	5/18/2025 12:28 PM	File folder
2025 chain hoist	1/14/2025 7:40 AM	File folder

Initiating an Assembly

An assembly file can be a parent or a child, regarding its purpose. In the below audio amp assembly, the design of the housing is the main goal while the amp and the assembly are created to perform the housing design. So the assembly design is a "child" of the design for the housing part.

The directory having the housing then also hosts the design of the amp and the assembly. So the folder has a purpose of one design: housing.sldprt. And the assembly is a necessary document to complete that design. Then, the necessary documents also live in the folder of the audioAmp.sldprt.



Fasteners – Nut Pocket

Hexagonal pockets are usually implemented for assembly convenience, and visual indicator. They are not expected to replace a wrench as a counter-tightening mechanism except for small amounts.

The most common sizes are tabulated. Note the second column is a nominal value, and final column is the designed size of circle containing inscribed hexagon. A depth of 4mm is a starting point for extrude-cut depth, where the nut does not sink out-of-reach in the pocket.

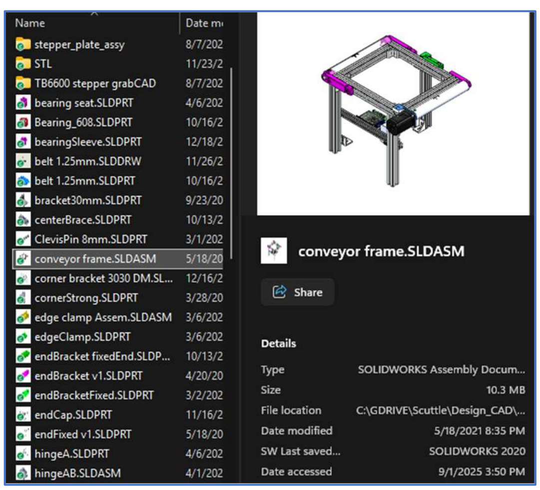
Metric	Hex (mm)	Design (mm)	Depth (mm)
M5	8mm	8.3	4
M6	10mm	10.4	4
M8	13mm	13.3	4

To be discussed

Assembly as Design

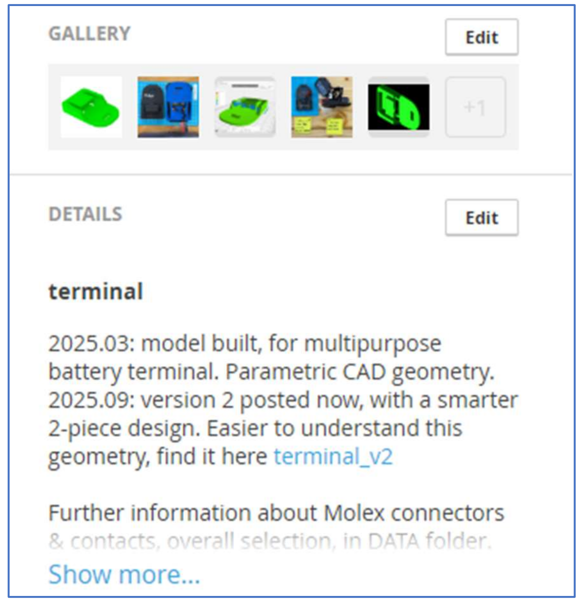
Some designs focus on the assembly and the assembly takes on a "parent" function. The goal is to design an assembly, not a part. Ideally, each part is finalized. The design effort is strictly separate for this project and the part-design-project.

In this case, each necessary part is copied or downloaded from its origin (web, catalog, past parts). The parts are placed into the folder of the designed assembly. A **problem arises**: parts become duplicated and lose searchability across the hard drive. Solution: when the assembly project is completed or paused, the workspace folder is zipped and the parts don't appear in other searches.



Parts Update / Upgrade

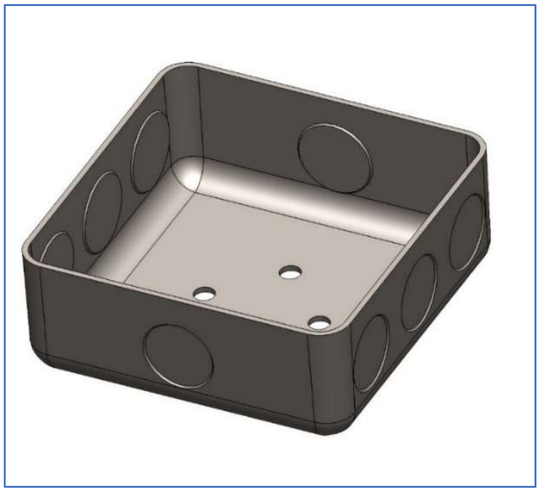
When a whole new redesign takes place for a part, then I post the updated version as a new post and include "version 2" in the grabCAD post. Then I go back to version 1 and edit the description. Include a link to my new version inside the description for the old one. At minimum, make a mention of the new version.



High Degree of Variation

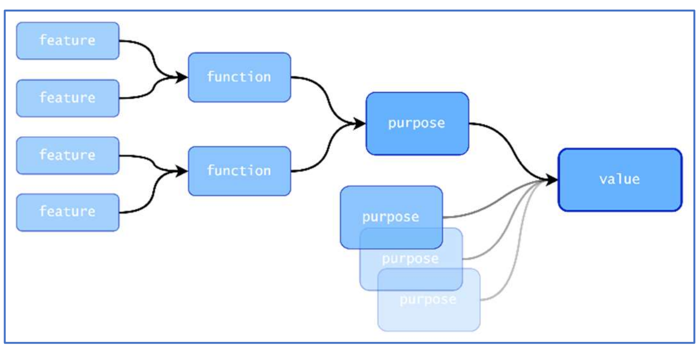
Some parts have parameters which contain more parameters and many variations that each have high-value applications. They quickly expand across multiple assemblies and projects and duplicate. For these parts the documentation required expands, and the community collaboration belongs at multiple levels.
The electrical outlet box called "handybox" by the Raco brand is one of these parts. The names and keywords become more complex, as redundancy appears across many documents. The image below shows a square-shaped handybox which is equally standard as the rectangular box.

When the part reaches this level of complexity the documentation requires more flexibility than a typical CAD model posting site. This part has been given a Github repository in addition to multiple grabCAD posts. The repository called "OpenBox_Project" receives a custom hyperlink at qr.net/openboxproject and several written explanations to describe the many functions that are under design development.



Feature, Function, Purpose, Value

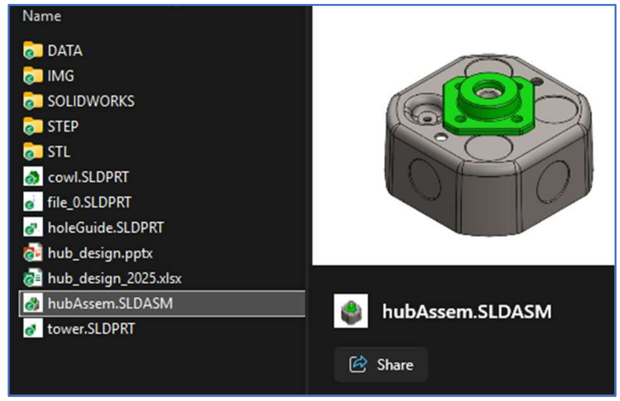
These terms are used in many ways but a more specific definition will help with discussions of shared parts, redundant parts, parametric parts, and parts with many functions. The diagram below shows the relationships: features of a model are formed to establish a function. Functions are assigned as a design goal for a part. Multiple functions may appear in a part for it to serve its' purpose. When we talk about simplifying and reducing designs, we must reduce all the way until a further reduction will damage the function.
The value term specifically can apply at any level in the tree. Features which support more functions are high in value. A design (at the purpose level) which also has many sister purposes (supported by the same design elements) has more value.



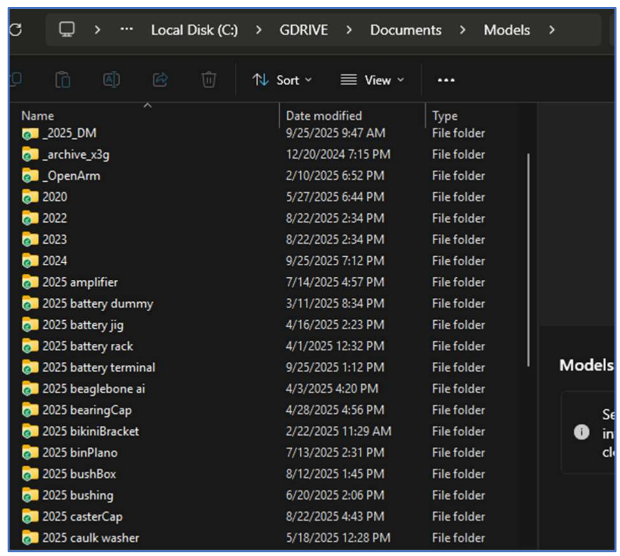
A representative feature is possible, or a representative function, purpose, or value. In the next two images, we aim to establish the high value of the square handybox by demonstrating it can purpose as a robust bit holder, or it can purpose as an airflow distribution hub.

One Part, One Design

Aim for one part that is designed, to achieve one function, and lives in one folder. Let's call this OPOD. For this publication and for everything up to 2025, we're addressing simple designs. To start the project, we make one new directory (folder) and give it the simplest unique name. See below, where the hub design lives in a folder, and that folder is for all information necessary to complete the design.



One level above, we have the Models directory where the other designs each have a folder. Again, note the simple names.



The next photo may intends to communicate a purpose in concept, before any CAD models are created. Actually the square handybox is already modeled and posted. At the moment of sharing the photo below. The two added components (pc fan and adjustable nozzles) have the functions of routing flow and directing flow, respectively. If someone is familiar with both of these parts, then they likely will ascertain the concept shown: we can use the square box as an airflow hub and easily design adapters to connect the components together to serve this function.

Copying Models:

A solid model is sometimes copied exactly and retained in a directory with a new assembly. This takes place when:

- The model is needed to make a new assembly
- The model has a low file size.
- The model has an original that may receive further adjustments.
- The model is completed to receive a version number.

Copying of a model is undesirable because the files can easily become confused. Future improvements can be accidentally implemented on a child copy, rather than the original. Future copies may be made of the original, causing improvements to get lost in a copy. However, we can take steps to reduce the mixup of files.
The minimal required step for copying a model is to add the suffix "COPY," and further steps can be added later.

This section may expand to a great deal of information so I will just include one example part: Aluminum extrusion, 30mm, which is used in SCUTTLE Robot and many other models. Consider projects which modify the value of the model

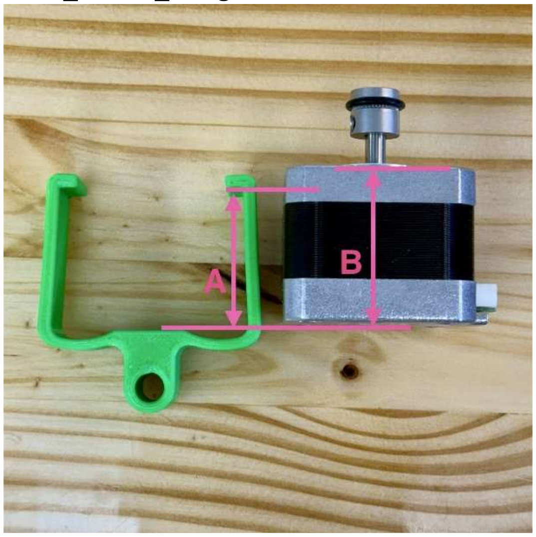
1. One project adds a variation of the profile to a sketch. The model becomes a repository for carrying geometry variations from multiple suppliers, with three configurations. This is essentially three models in one.
2. One project minimizes the detail of the model to produce a lightweight copy – easier for including in large models and simpler for newcomers to use for mating in assemblies.
3. One project takes the official McMaster supplier's CAD model and rebuilds the model by excluding advanced Solidworks Features. This one is best for inclusion in assemblies that are posted publicly.

Among these projects, the designer must choose which to copy for a new assembly that requires this material. The action is to make it clear which project is associated with each model and ease decisions when a future project inevitably causes a copy. Perform this action by:

- Add a note.txt file to the directory containing the models. Decouple this information from the model, so the key information is not lost in the case of a model overwrite.
- Add an unusual name for the models which are not the default for copying, such as "Extrusion_VariantA.sldprt" instead of Extrusion.sldprt.
- Create a disambiguation assembly which places each variation side-by-side, then use this assembly to confirm names, perhaps adjust material color, or make other minor changes that make the difference noticeable in the future.

Compliance: Strong

To impose a strong force in an assembly, the compliance has one method and a weak force uses another method. The strong compliance requires a more precise dimensioning for deflection. The deflection comes from an interference in the design-part and the subject-part. For this deflection it will most likely need repeated trials but the desired range is 1-2mm.



Sigma Compliance

Use sigma (**Σ**) compliance for gripping symmetric parts. This method adds more tolerance to a design for mating two parts without fasteners.

In the above example, the green part has compliance in two key directions for deflections in a “weak” and “strong” directions. The rigid finger tips clamp while the arms pull (left-pointing in image) the motor to the back panel. The arms also deflect outward for a pushing force towards the centerline of the part, and this is secondary. The secondary deflection is only sufficient to close the gap between arms and motor sides.

Background

Drafted discussion not yet integrated in the document.

I'm nearing a point where each new CAD model takes very little effort, and I'm just producing the design from a set of rules. I'm making very few decisions. That means this is becoming systematic. A signal of proper engineering.

What is a software language? It's a system of syntax and rules to write instructions that a computer can interpret and execute. A software language provides a bridge between human thought and machine execution**

What makes a software language complete? (if I were developing C++, how do I know if it is finished?) There is an objective answer, and the most common reference is TURING COMPLETENESS: a language is complete when it can, in theory, perform any computation that a turing machine can, provided enough memory and time.

Is it possible to make a *mechanical set of elements* that covers all of the necessary components? If it's possible to make a complete textbook then it's possible to define a complete physical set of solutions. What makes this texbook (Mechanical Engineering Design) complete? It's a book for mechanical design. I think its when a collection of features can be formed into (nearly) any mechanical machine.

So if I can describe the system, describe what criteria will make it complete, then get help from other designers, then we can achieve the reinvention of everything.

The difference between these two parts is 3 floating point values. THREE! The hackers out there should be able to modify my CAD files in a matter of seconds - and then autonomously, thereby making each design into 1000 designs.

The CAD problem is now solved, in the way mathematicians write a proof. I built and released enough examples to demonstrate each feature and I want to ask for help.

Completeness
I am near a point of completion; of having a small set of functions implemented that enable any possible mechanism that can be desired.

Materials-Structure
1. plastic (abs & PVC)
2. aluminum (extruded 6061)
3. steel (fasteners, pins, balls)
4. steel tubes, EMT conduit, 1/2in and 3/4in trade size

Materials-Panels

1. Expanded PVC, 1/4in thick, panel
2. Foam sheets, around 25mm thick, porous, light compression
3. Rubber panel sheets, 1/4in thick, high-strength
4. Composite pegboard or smooth board, 1/4in thick

Materials-Laminate

* vinyl wrap, 12-in rolls
* epoxy, 2-part, odorless*
* cotton duck-cloth (like canvas, durable, flame-resistant)

Materials-Geometric

The maximum reduction of features leads to highly characterized parts
* steel round balls, bearings
* round tubes, copper
* round pins, wood & steel
*

Engineering-types

Product-Types
Parts function expressed by product names & trade names
1. one way valve
2. machine screw
3. structural tube
4.

Function-types
We need a rule to describe the meaning of "function" and it's relationship to a "part" since some parts can have multiple functions, and so on. This note is a placeholder to verbalize my rules for scale and spread and discretion between functions and parts.
1. handle
2. joint, load bearing
3. knob, turning
4. leveling foot, adjustable
5. wheel, free-spinning
6. four-bar linkage
7. structure attachment, tension
8. structure attachment, sliding
9. structure attachment, pin on plate
10. structure feature, plate mating

Rules

1. Material introduction: A material can only be introduced if accompanied by a method of permanent bonding to all other materials in the collection.
2. Fasteners: A fastener cannot be introduced except accompanied by the method to perform the fastening and the method does not expand the tools required in the tools collection.
3. Parts: parts must be assigned a tier where the tier 1 parts are the most simplified and natural form of the part and the number of parts in tier 1 is to be minimized. Tier 2 parts are those which can be described with the same definition as a tier1 part but having more specific details. (ie bearing at tier 1 may feature a family member "sealed bearing" at tier 2) The collection of parts will reach a finite limit and be expressed by the Tier1 parts overall.
4. Parts introduction: A part cannot be introduced which has a function already accommodated by an existing part in the collection. An introduced part also must include a standard method of integration and a most favorable example of integration which expresses the purpose of the part. The example of integration is to map to an example of a purpose of that integration which has been introduced at or before the part's introduction.
5. Materials Expansion: A new material is not to be introduced if the function can be achieved with the existing materials in the collection. A new material may only be introduced as an end-node in the assembly of a machine. The end nodes of a machine are that which can be interchanged without changing the design of the machine.
6. Parametric-OTS: Parts may only be added if they originate in a family which features variations of the key specifications. A tube is sold by length, then the length parameter must be variable continuously.
7. Metric-first: metric selections of parts must be used except if they operate on a supply stock based on discrete imperial units. The slotted Unistrut channel features discrete repeating geometry at two inch lengths and with 1/2inch holes, so the discrete inch values can be computed in metric easily without expanding the mathematics tabulated in our design files.
8. Permanent Joints: Permanent joints always accompany a method for joining which is defined with the minimal supplies and geometric features necessary. with a method such that future jointsmust integrate the same joining method that is best-qualified to-date unless the joining method is not suitable.
9. Permanent Joining: the permanent joining

Rules-Bonding
1. Joining is divided into PERMANENT and TEMPORARY bonds.
2. Example Piece: web.sldprt
    1. The bonding uses a compatible adhesive, chosen before design of geometry
    2. The mating parts are constrained in all DOF except for one, before using adhesive.
3. Adhesive only constrains 1 degree of freedom.
4. Adhesive is planned to carry load in shear-direction (not pulling)
5. Ideally, no priming. Use primer if performance isn't achieved with pure adhesive

Rules-Modeling-Features

1. A model should use the fewest dimensions to form the concept.
2. If the concept grows to have an added function, (see example: bearing & pvc) there must be made an additional model.
3. The model matures when it can gain new functions by adjusting only one dimension. The adjustment of this dimension becomes the utility of the part. In the parametric bearing, I can purchase a new ball, input the new diameter, and the model gives a new design for the bearing size, which can export to STL and make a part. All further variations become a new model with new descriptions, and should note their "root model" from which it started.
4. Implement symmetry to reduce the complexity of a model. In many models the main body is extruded up from the top plane. The model is improved by returning to the first sketch & extruding from sketch as "midplane." This has a cascading effect on models, adding far more power to constraints like "symmetric" and "equal length."

To finish this project

1. must finish the collection of templates. To logically check if the collection is complete we can use the machine-planning concept. Imagine a desired machine to be built with the collection. The machine should be able to be reproduced with the collection at-hand. If it cannot, we explore if a simpler execution of the machine exists which still achieves the defining function. If we find a simpler machine, we displace our present selection. We continue reducing the machine concept (performing machine reduction) until we find the simplest execution. Then we review if the final selected machine can be built with our collection and if it cannot, we ask what portion is not possible.
2. If it cannot be built with the collection then

Reduction-Efforts
* Use a template material for *

Goals-Hoshin

Particular goals are continually applied, continually improved, and never quite complete. These goals are strong enough to become rules but they lack a discrete (yes/no) validation method.

**

The apparatus shown above is an example design which has the **minimal necessary parts** to demonstrate the function of the selected water pump. We need to be able to answer all the questions in our examples:

- What is the flow rate (with flow that is visible to the eyes)
- How much power is consumed by this scale of a pump?
- How much control do we have to adjust the flow without changing the actuator selection?

same

as having as having impulsive decisions, but such description is oversimplified. Instructors face constraints on their .

Closing Remarks

Be advised this guide is being written over a period more than 1 year and is greatly influenced by feedback from readers over time. After 10+ years of 3D printing, the knowledge moves into the “intuitive” mind and this paper is a journey of laying out subtle habits in clear writing.

I am hoping that we can build a world where designers can perform less effort rebuilding what is already built and that we can become experts in exchange of valuable data such as parametric designs. This only happens by communicating with each other so you are encouraged to reach out with inputs.

For students, I hope that you can utilize my example parts just like you would use a fastener, off-the-shelf, but in a new way. The fastener was refined over 100 years but these parts are only a few years old. Your inputs and improvements can make a difference.

Find this paper and my example parts listed at qr.net/openlabproject with many more resources.