James Kesler
james@kenazdesigns.com
509-715-7101

Cad Design guidelines

Years ago when I took a basic self-defense course, the instructor told the class, “The first rule of self-defense is don’t be stupid.” These guidelines are meant to inspire thought and discussion, and help bring some consistency to part designs. This will help you and others revise your files easier.

That being said, if following a particular guideline is causing you fits, creates unstable models, or promotes complexity with no benefit, “don’t be stupid.”

One other word of caution, this document is intended to be a living document. I will revise it at will as I come up against problems that could have been avoided.

Finally, this document was written from the perspective of a long time Pro-Engineer/SolidWorks user. It has application to other CAD systems, but some terminology or feature creation methods may not line up.

Design Intent

Design should follow intent. Dimensions and features should reference features and locations on the model that follow the intent of the feature relations.

Features should be named in the design tree. This will ease any future modifications whether you do them or someone else.

External References

External References and Assembly features should be used sparingly, if at all. The legacy of such features can have real unintended consequences down the line. If the models that use these features are re-used outside the design, the problems are compounded by unintended changes.

Even if the parts and assemblies are only used on one assembly and not re-used on any other, external references and assembly features can cause revisions to impact more than what was intended.

No matter the time savings these features can provide, experience has shown that these features often cause more churn and time wasting than they save.

Instead I’ve found that allowing dimensions to be modified on drag helps align sketches to mating parts more easily.

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Decimal Places

Try to use round numbers rather than 6-10 decimal place numbers on dimensions. It speeds checking dimensions between parts and makes drawings cleaner.

Only use decimal places you need. I know of no machine shop or manufacturing process that anyone besides the government can afford that can produce a part to 5 decimal place accuracy. Why design to that accuracy?

Sketch Creation

Choose faces or planes that relate to part design intent for sketch creation.

Use fully defined sketches. Use the correct method of constraint. Adding dimensions to points to constrain entities to one another is never allowed. In SolidWorks, I have the fully defined sketches option enforced to make sure this happens.

Sketches should be as simple as possible but as complex as necessary. By making a sketch complex, the number of features can be cut down, but straightforward modifications are often prevented. Sketch complexity vs feature count should always be kept in mind.

No “magic dimensions”. Magic dimensions are used to fully define sketches or features that can’t otherwise be constrained. They do not follow feature intent, and are not tied to geometry. On part re-size, magic dimensions will often cause model failure because they don’t re-size with the geometry.

Machined Part Design

No square inside corners on finished parts. This goes more to manufacturability but is driven by CAD geometry, and can be most easily fixed in CAD. Machine shops can create square inside corners when necessary, but the process is complex and time consuming. This leads to additional and often unnecessary cost.

Generally when modeling machined parts, it is preferable to create a block extrusion that encompasses a majority of the final part volume and use extruded cuts and etc to make the part. Form follows function, and creating geometry in this way will help the designer keep the machining processes and manufacturability in mind as they design features. Once again, design should follow intent.

Screw machine and turned parts should be made in a similar fashion. I like to start with a nominal rod diameter for the part and then use revolved cuts as much as possible to create the desired geometry. This relates to the primary method of material removal used in screw machine operations. Any live tool operations on a screw machine part should follow the rules above.

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Molded Part Design

Use of the shell feature really simplifies design, and makes the part nominal wall more consistent and correct. Any feature that impacts the shell (has an effect on both A and B sides of the part) should be created before the shell if possible. Some features cause issues with shell

Radii along drafted walls should never be drafted. Rather the radii should be added after drafts. Otherwise the radius will be variable along the length. This will cause cnc programming issues and a lot of tool changes. Variable radii are almost as bad as square inside corners.

As a general practice, a part with a lot of radii seem to be more stable if vertical radii are added first and horizontal radii are added second.

Any shutoffs should be made planar. SolidWorks has a tendency to warp shutoffs into non-planar geometry. This either causes more work or more complex cutting and burning when the tool is created.

Sheetmetal Part Design

Stretch and Rip relief should never be used. They are very difficult to impossible for sheetmetal houses to replicate in production. This will add cost. Or, they will be modifying your parts to fit their capabilities. Modifying parts downstream can lead to mistakes.

More to come in future revisions

Assembly Constraints

Multi-body parts

Use of configurations

Revision Table

REV	Date	Changes Made
1	21 Apr 2016	New Release