Good Modeling Practices

Rediscovering "Good Modeling Practices"

Regardless of the CAD tool, design intent and choosing and managing the proper references are the key behind good parametric 3D modeling. Choices in how a model is created make a difference in how easy it is to manipulate, extending its use, and the ROI of the CAD tool and user training. Modeling techniques should be used that enhance the usability and robustness of a model. As you create features and models, think about how easy or difficult it will be for you, or more importantly, others, to manipulate for future changes, release, or leverage for the next generation of product. Take the time to build flexibility into your models, and choose good stable references that reflect and drive design intent. Chances are very good that other people will end up working with your models. Because there are always circumstances that change the relevance of the rules, there are no hard and fast rules for good modeling. Rather principles and guidelines.

Below are some best practices and guidelines. They are by no means complete, but contain things that can make a difference in the usability and robustness of the typical model. There are some redundancies between the Best Practices and Guidelines, but the points are the same.

Keep in mind that the material below is not set in stone. They are guidelines and good practices, but in no way absolute.

Following guidelines and best practices can appear to take a little longer upfront and impact deadlines. However if practiced and adhered to, changes, product release, or leveraging for a new design will be seamless and timely.

Recommended Best Practices

Always start with a plan

Before beginning, have a modeling plan or strategy in mind. It should incorporate the design intent of the model. It's not necessary to have it fully developed before beginning, but having a plan is important as it will undoubtedly change as a project evolves. It can be easy to change the wrong plan into the right plan. It can be very difficult to change no plan into the right plan.

Use Company Standard start parts

They contain default datum's, layers, parameters, views, etc and will make life much easier as your models, assemblies and drawings evolve. They provide consistency

and fulfill the minimum expectations required from other users.

It’s all about design intent

Design for "product design intent" first and foremost. Create sketches to ensure they are constrained the way you want and react to changes as you would expect. A majority of problems have to do with poorly constrained sketches that don't accept change easily.

Create and choose references (parent/child relationships) that drive design intent

This is a fundamental concept that is not always properly understood.

It’s important to understand that every time a reference is chosen, a dependency, or relationship is created.

When creating features, of any kind, always be aware of what is being referenced. Pay attention to the command line as it tells you what you have chosen.

Don't readily accept the default references a CAD tool gives you. Think about the design intent and choose references that are stable and reflect that intent. Automatic references do not always reflect intent and are rarely applicable or appropriate.

Choose references that allow features to "move" with the intent as changes are made.

Choose references that won't disappear

References like edges, that disappear when rounded, are usually not the best choice. Datum’s and planner surfaces are typically better.

Choose references sparingly and wisely

More references mean more feature inter-connectivity which can make a model more difficult to work with. At the same time choose enough references to make the model follow design intent. Some models are so tightly tied with references that they fail with almost any modification. Users wonder why parametric tools are so difficult to work with, but more often than not, it's because of the way models and assemblies were constructed.

Use datum features to your advantage

Use datum features (planes, axis, points, etc) and external sketches to help define intent. Creating these types of features, even prior to any solid geometry in a model, in order to drive intent is perfectly acceptable. External sketches can be used to create a part “footprint” that can be used for later features to reference.

When several features are to reference the same thing, create datum’s for control

Key datum’s (planes, points, axes) are easier to find and select when they are named. Named features also denote significance for someone changing the model later.

Build flexibility into your model

Features/models should be created in a way that allows other users (who may be unfamiliar with your model) to readily (and successfully) modify them at any time. Good modeling practices are far more important early in the model than at the end. It’s a reasonable expectation that future/ongoing design changes should be able to be made quickly and easily by people other than the original designer.

Keep sketches simple

It's usually best to have many simple features rather than a couple of difficult ones. Don't try to create complicated sketches, but break them down to the parts core shapes. This will make it easier for modifications down the road.

Create features individually (by not creating what should be multiple features as one)

Avoid creating one feature that should be multiple features. This will allow any combination of features to be suppressed or deleted. If what should be multiple features are created as one, you are doomed to an all or nothing scenario. Create features in a logical sequence.

Create relations that can help to drive intent and associate features when direct references are not practical When writing relations, use comments for "what" and "why". If others work on the model, their life will be easier if relations are commented (i.e.; explained) **Drafts, rounds and chamfers are often best left to the end of the model**

There are always exceptions, but in general, care should be given to where these types of features are inserted in the model tree. Avoid using these types of features as references for later features. **Drafts, rounds and chamfers should normally be inserted as features rather than put in a sketch**

Again, this is a good rule of thumb, although there are times when these feature types can't be created independently or a dimensioning scheme (design intent) requires it in a sketch. Its good practice to build models without drafts and rounds then going back and using Insert Mode to put them in, (along with grouping them with the parent features). **Robust and stable models (error free)**

Ensure models (parts and assemblies) regenerate without warnings and errors. Run model check (or other useful diagnostic tools that are available) often.

Address and resolve any model failures. If you understand the concept of Parent/Child relationships and feature dependencies, you can almost predict when a model will fail. Learn how to resolve failures and it will help tremendously. Bypassing failure mode, possible in a number of CAD tools, is fine during the design stage, but should always be resolved. Make life easier for the next person.

Create and use mapkeys

Learn to create mapkeys for commands that are both frequent or require multiple menu picks (like exiting sketcher, erase not displayed, show/erase, make note, etc). Incorporate mapkeys into your daily routine and productivity will increase dramatically. Your mouse hand will appreciate it.

Use GD&T

If you don't know it, learn it, inside and out.

Show model dimensions in drawings

If you create drawings with dimensions, use the dimensions that were used to create the features/model by “showing” them. Avoid manually creating them.

A model and the object in a 2D drawing view are one and the same. Manually created dimensions can not be modified in a drawing and don’t exist in the model. If a change needs to be made to a manually created dimension, something else will need to change in order for the created dimension value to change.

A common reason for manually creating dimensions is to get a desired dimensioning scheme, however if models are thought out and built well, redefining can be quick and seamless.

Lastly, changing the number of decimal places in manually created dimensions does not regenerate the model resulting in differences between the model and the document defining the model.

Guidelines/checklist to good models and drawings:

General

- Use your head. It's the best tool you have. Think first, and then model
- Use start parts & assemblies for all new models
- Let design intent be your guide

- Use Parent/Child references to your advantage. Don't just pick anything or whatever’s convenient
- Do not create unintended parent-child relationships. Pay attention to the references you choose
- Use datum features (planes, curves, axis and points) to help define/drive intent
- Learn to create datum’s-on-the-fly
- All applicable parameters should have a value assigned
- Clearly comment all relations
- Use Layers to control the visibility of models
- Do not use family tables for anything except a true family, such as fasteners
- Don’t add material only to remove it later. Don’t remove holes by filling them with material
- There are often multiple ways to create most features. Think & choose carefully. Different feature types behave differently when modified. Think of what’s likely to change in the design cycle and base your choice on what would be most appropriate.
- Think like a machinist when creating machined parts. It can help to follow and understand a design.
- When designing in a concurrent engineering environment, use shrinkwrap or envelope parts from a model for use in upper level assemblies. Bad geometry created at the part level makes creating lightweight parts harder.
- When importing models from outside sources (IGES, STP, STL, etc), import into a company start part instead of just opening.
- Use mapkeys to increase productivity
- Learn to use and manage layers. They will be your friend down the road.
- Put as much information into your model as possible (gtols, datums, sections, views, etc) instead of in drawings.
- Model around ASME Y14.5 or whatever standard you're required to conform to for dimensioning and tolerancing. Use realistic tolerances. Pay attention to number of decimal places, they matter.
- Save early and often

- Use built in diagnostic tools (like Model Check) to find geometry related problems like very small edges.
- Resolve errors and warnings (geom check, circular references, etc)
- Do not save a model with suppressed or incomplete features.
- Use the erase option to clear objects from memory

Sketcher Mode

- Use default datum planes to orient sketches. Sketcher orientation references are the most overlooked reason for feature failure. This should almost always be a default datum to avoid possible problems.
- Make sketches simple. Complicated sketches are more likely to fail and take much longer to regenerate. This speeds regeneration time and makes modification less painful. The number of features used is irrelevant. A complex part can involve simple sketches.
- Think about how a design may change as it evolves and build that ability into a sketch.
- Use a combination of dimensions and constraints to convey design intent.
- Use sketched centerlines, construction circles and points to help achieve the desired dimensioning scheme.
- Dimension sketches as needed in order to ease the use of showing dimensions in a drawing.
- Align sketches to surfaces or datum planes whenever possible. Try to avoid using edges.
- If the "use edge" option is chosen to create a sketched entity, un-align it from the edge and realign it to a surface.
- Avoid weak (grey) dimensions as they can cause problems. Design intent should only contain dimensions and constraints required for the feature. Weak dimensions that can’t be gotten rid of often indicate some sketch problems.
- If possible always use REPLACE (under EDIT) instead of deleting a sketched entity. REPLACE will give the new entity the same id number as the old entity. This will result in rerouting all the children of the old entity to the new entity.

Part Mode

- Choose references that follow/drive design intent.
- Avoid referencing a model edge if there is a suitable surface available.
- Create datum’s for controlling several features that reference the same thing.
- Use datum planes to control depths or heights where many features will have the same value.
- Do not create material only to later remove it. Delete unwanted holes and cuts instead of filling them in with material.
- Features should not be covered completely by other features
- Create drafts, rounds and chamfers last when possible
- Add chamfers, drafts, and rounds at the end of the model.
- Avoid creating children of chamfers, drafts, and rounds.
- Avoid creating rounds and chamfers in sketcher; they should be added as separate features when possible.
- Use patterns whenever possible. They should be used in preference to copy (although there is a time and place for copy)
- Rename features and dimensions so they can be logically understood and more intuitive (such as "Parting surface").
- Use Local Groups to group related features.
- Avoid suppressed or incomplete features
- Use thin sections when applicable
- Always keep the same parent surface for merges.
- Fix missing reference warnings
- Check for and remove "geom checks". Geometry Checks should be grayed out
- Orient views using default datum’s or view names, not model geometry
- Use Relations to build interdependencies among features and to drive design intent (such as preventing a radius from becoming too small or too big). Make sure you comment your relations.
- Let manufacturing guide whether you choose to create a Hole or a Cut with a circular sketch. A sketched hole can easily change to a slot where as a hole and sketched slot are two different features.

- Use Make Datum’s rather than external datum planes if no other features will ever use that datum.
- Transform surface features instead of Copy-Mirror or Copy-Move. Transformed features regenerate faster and decrease file size.
- Don't delete and remake similar features. Always try redefining first.

Assembly Mode

- Assembly features, such as cuts, should be avoided unless necessary.
- Use constraints that reflect real-life assembly or manufacturing situations
- Follow the rule that assembly datum’s make the best references (parents)
- Do not select constraints simply based on ease of selection of references
- Frozen, suppressed and/or suspended components should be resolved.
- Don't use external references unless necessary.
- Create explode states in assembly mode and “set” them in drawing mode.
- Use Simplified Reps to control the visibility of components in an assembly.
- Use coordinate system to assemble a first component.
- Avoid “packaged” (unconstrained) components.
- Avoid “offset lines” unless necessary (showing axis in drawings works better most of the time)
- When using family table parts, always use instances in assemblies, never the generic.
- Use hardware from the company standard Library.
    - All models have defined parameters (to drive BOM info)
    - Reduces redundancies
    - Saves time

Drawing Mode

- Use only company standard formats and templates for drawings.
- Use only named views, created in the model, in the drawing.

- Do not manually create dimensions. Use “shown” dimensions (dimensions used in the model to create features) as much as possible.
- Avoid "faking" dimensions and tolerances in drawings.
- Drawings should reflect both design intent and manufacturing needs.
- Drawings should not contain overwritten dimensions (in the case of ProE, @o).
- Models attached to drawings, but not showing in any views, should be removed.
- Datum visibility at the drawing level should be controlled by hiding all datum layers.
- Correct any errors upon regeneration
- Avoid erased views in drawings
- Avoid “draft” entities unless necessary.
- If draft entities exist, relate them to views and/or objects so they move when the parent view/object moves.
- All title block information should be driven from model and PDMLink parameters
- Use company standard balloons
- Avoid phony BOM’s and balloons
- Create explode states in assembly mode and set in drawing views
- Create sections in part/assembly mode and set in drawing views
- Avoid sectioning hardware
- Pay attention to hidden line display (more often than not, no hidden lines should be set for views). This is especially true for ISO and non-orthogonal views.
- Remove hidden lines for any quilts (objects created as surfaces)

Sheet metal Mode

- If it’s to be manufactured in sheet metal, model it using sheet metal.
- Flat patterns should be for reference only. Sheet metal shops have their own tables for their k-values. A flat pattern created in ProE or SolidWorks may not necessarily match theirs. Sheet metal shops are flat pattern experts. Leave the flattening up to them and only dimension the final product.

- Three place dimensions for holes and other important features only, two places everywhere else. The tightest Wall to wall dimension that can be held is close to +/- .02" at best.
- Avoid sharp corners. Chamfer and/or round sheet metal corners.