

3D Modeling System Best Practices

The following document includes a set of general best practices for model creation. These practices are not software specific but appropriate for any of the professional grade constraint-based systems such as PTC Creo, SolidWorks, NX, Catia, or AutoCAD Inventor. Use of these practices will facilitate the creation of files which are robust, easy to edit and appropriate for use with downstream applications such as CAM, CFD, FEA and additive manufacturing.

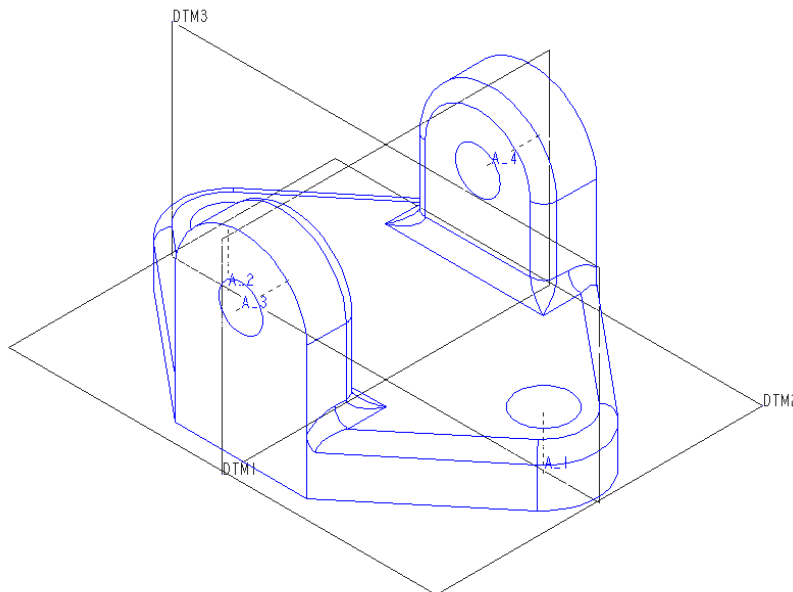
Model Scale

All models should be created at actual full size (1=1), regardless of whether using Metric or English units.

General Model and Assembly Orientation

Plan the model orientation with respect to the software default isometric or trimetric orientation such that the most descriptive view is shown. Use primary datum features to orient the component. Consider symmetry and hole positioning when orienting the component. As a rule, the horizontal plane is XY and the vertical direction is Z.

When creating parts for an assembly, use the same construction orientation for each part you make. Also consider using datum features as reference geometry in assemblies (mating and aligned features).



Model Creation Practices

General Creation Practices

The primary features of a model (first created, if not default) should be a set of three mutually orthogonal datum planes.

Production models should be created with parametric features, but this requirement is not applicable for the commercial standard / common (read purchased) parts used in final design assemblies.

The required parametric modeling techniques must be robust (no error indication occurs when any parameters are changed). This can be accomplished using horizontal modeling methodologies. Un-parameterized solids should not be used within a project, with exception of downloaded files of standard / common parts.

Changes (corrections) shall be made to the model features by editing parameters of those features when possible, not by adding new features.

When created several components for an assembly, consider the use of master (or startup) files. Standard settings will be defined in the master files. Examples are units, mass property output, drawing settings, etc.

As a rule, a model consisting a multiple, simple features is easier to edit and hence more robust than one created from a few complex features.

Models

When possible, new features shall be referenced to master layout features (such as datum features) instead of prior features [Horizontal Modeling Principals].

When choosing model colors avoid using the software default geometry, construction and selection colors.

Models will be created as efficiently as possible, using solid feature operations (holes, slots, fillets) whenever possible.

In general it is most efficient to create a component model in a sequence that mirrors fabrication.

Any part with the primary physical shape of cylinder or sphere must contain appropriate datum features to facilitate assembly. An example would be two, orthogonal datum planes whose intersection defines the primary axis of the cylinder.

If more than one surface is used to define a lofted feature, the adjacent surfaces shall have a minimum of C1 (tangent continuity) to each other; C2 (curvature continuity) is preferred.

Sketcher

Sketcher should be used to capture the design intent of the primary features.

Secondary features such as chamfers, straight holes, rounds and fillets should not be created within the sketch but should be defined and added using feature operations.

Sketches should be fully constrained with use of appropriate dimensional and geometric constraints.

Sketches should be organized and kept as simple as possible.

Relations

Relations, arithmetic or conditional statements that have a value, may be used to control dimensions and relationships between features and parameters in the model.

Expressions

Expressions, equations or conditional statements shall have proper values and units, and should be used to control dimensions and relationships between features / parameters of the model. Expressions that control Key Parameters of a design feature or assembly shall utilize a user defined expression to facilitate understanding and conveyance of the design intent.

Key Parameters include, but are not limited to, the following types of information:

- Parameters that control the overall size of a model if applicable.
- Simple sheet metal (consistent material thickness throughout) parts require the definition of material thickness.
- Parameters that are expressed using other dimensions or using engineering formulae.
- Parameters that will be used as a basis to generate families of parts, design changes, important mechanical interfaces, etc.

Feature Creation – order

The order of feature creation is critical in successfully capturing design intent. Model features should be created as follows:

- 1) Begin with a set of three mutually orthogonal datum planes.
- 2) Create the exterior or primary features first, in a logical order. These features carry the general shape and intent of the part.
- 3) Create interior features next, in a logical order, following the appropriate primary feature(s). Usually these are interior cut features of the primary geometry.
- 4) Create secondary features, i.e., holes, draft, rounds, in a logical order, so that the purpose of the design is fully met.
- 5) Create cosmetic features, such as symbolic threads, in a logical order, i.e., create the hole, the chamfer, then the symbolic (cosmetic) thread.

Note: The creation of holes using cylinders or sketch circles should be avoided.

Chamfers, Holes and Fillets

Chamfers, holes and fillets shall be added to the model using dedicated Feature Operation commands and should not be included in the sketch of swept or extruded features.

All hole features whose design intent requires a bilateral tolerance (hole size with a +/- tolerance) should be created at the basic size.

All hole features whose design intent requires a unilateral tolerance (hole size with upper and lower limits) should be created at MMC (the minimum diameter)

Holes should be positioned using drawing information, that is reference the actual datum features to be used in fabrication and control..

Multiple planar holes with a regular pattern should be created using an instance (patterning) feature operation.