ME 24-688 – 3D Part Design

Part Modeling Best Practices

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1 Introduction

1.1 Best Practices Introduction

This document will serve as a guide for designers to best approach 3D part modeling. These practices will assist in creating manageable parts.

2 Parts

2.1 Sketching Best Practices

- Project the part origin or some other RELIABLE and LOGICAL geometry into each new sketch and reference new sketch geometry to it.

- Sketches should be fully constrained and related to the sketch origin. Avoid using the fix constraint, it makes the sketch difficult to edit by another user. Create the base sketch around the projected origin, use reference geometry to constrain it and Extrude in either direction if the first feature is an extrusion. If the first feature is a Revolution make sure the centerline is constrained to the project origin and uses the Centerline Line Type. Thoughtful positioning of the first feature will give you more usable work features later and will be even more beneficial when you start assembling components.

- Apply sketch constraints and dimensions carefully and logically so that the sketch geometry will change in a predictable manner when a dimension is edited.

- Avoid placing fillets and chamfers in sketches. If possible, make them the last thing added to the part. Only add them earlier if there is a functional necessity.

- Use construction lines to make sketch relationships easier to analyze.

- Use cross part sketches only for strategic relationships. These are powerful in creating associativity across parts but can add complexity to making changes in your design.

- Use silhouettes only when absolutely necessary. These are highly depended on the sketch plane and its orientation to the source geometry. Large changes can have unexpected results.

- Turn off “show constraints” when done. The visibility setting of constraints is remembered and you might surprise someone when they go edit a model of yours, activate a sketch and all the constraints appear needlessly.

- When importing DXF and DWG (remember you can copy and paste) data use the Auto dimension command. Uncheck the dimension optional and then press the apply button until the number of constraints needed stops changing. You might need to click apply three times for this to happen. Now you should have a much better constrained sketch to start with and need fewer dimensions to finish fully constraining you sketch

- Less is more. Don't overload a sketch too much. Think 20 sketch lines and 3 to 10 dimensions per sketch. This helps break the model up into manageable pieces and makes editing the design easier.

2.2 Model Features Best Practices

- Rename key features in the feature tree. This makes it easier for another user to find and edit features.

- Use user defined parameters for common dimensions in a part where applicable. For example, if a typical radius is to be used in a casting design for fillets, define a parameter called "StdFilletRadius" and assign that value to applicable dimensions. If during the design process a universal change in the radius is required it becomes a simple change of one parameter and hitting the update button. This can also make it easier for downstream users to quickly identify the key design parameters.

- If the parameter has a value restriction (only + or - 1 or 0, 90, 180, or 270), make sure to describe the proper use of the parameter in the comment field. If you use an external source, such as a spreadsheet, to feed parameters to your model, make sure to note both the spreadsheet and model so that they reference one another. Add tolerances to model parameter if you know them.

- Use equations. If the design is quite complex, use the Engineer's Notebook to document what/how/why you've designed the way you have

- Avoid parent/child relationships between features. Unless necessary, avoid starting sketches on part faces or projecting feature geometry. Instead, use the origin geometry or work geometry based off the origin geometry. Critical relationships between features should be obvious and logical so that changes to other parts behave in a predictable manner. Objects to avoid creating relationships to:
    - Faces and edges of chamfers and fillets
    - Features in a pattern or mirror, unless it's the parent feature(s)
    - Edges or seams of non-analytic surfaces (like swoopy curvy surfaces from loft)
    - Non associative cross part projected sketch geometry that is fixed or grounded
    - Grounded work geometry

- Name your features if they are critical or commonly edited. Naming construction geometry will make working with the model significantly easier and should be done as often as possible.

- Add cosmetic features like decals and embossed text at the end of the feature tree.

- If a group of features are all relative to a point other than the origin, or at an odd angle to the origin planes, create a User Coordinate System (UCS)

- Avoid unnecessary features and work geometry. They increase file size, clutter the feature tree and slow down the program. This is important when working with complex parts. Strive to have a clean, efficient, stable and logically ordered part as your finished product. Downstream users

(sometimes, that's even yourself) should be able to understand and edit it as easily as possible. A minimal investment of time and effort during model creation will pay off greatly downstream.

- Don't take shortcuts. Edit features to make changes, use grips to help speed up finding the feature and dimension to change. Common examples to avoid:
  - Do not fill in holes to remove them. Delete the hole and repair if needed.
  - Do not stack extrudes on top of one another to make a part longer. Edit the dimension.

- When possible pattern features rather than sketches. These are easier to edit and understand

- Use placed features (Holes, Fillet, Chamfer) rather than sketching and extruding, revolving or sweeping.

- When making large sweeping design changes to a feature, drag the EOP to right below the feature you are about to change. Then progressively roll the EOP down and repair as needed. This approach is faster and easier than letting the model fail massively. If the model does have massive failures re-read these tips and try to create a more solid model and as you repair it.

- When filleting difficult models, turn off chain edges to reduce the number of edges that you are trying to round. Fillet corners with mixed radii and convex/concave solutions first then finish the other edges after.

- When e-mailing a native Inventor part, roll the end of part marker to the top of the browser and save the part. This will decrease your model's size

- Unless a work feature is critical for downstream use turn visibility of each individual work feature off after use in the part.