Top-down Assembly Modeling using Control Models

A systematic way of controlling inter-part references!
Bottom-up versus Top-down

• Recall that in **bottom-up** assembly modeling individual part **solid models are created first**, then mated in assemblies.

• In top-down assembly modeling the **top level assembly is created first**, then components are added as empty part or sub-assembly files. The part solid model geometry is created last. They are modeled “in context”.
Examples from Industrial Bolting Technologies

MAE 455 Computer-Aided Design and Drafting
Bottom-Up vs. Top-Down Assembly Modeling

**Bottom Up**

1. 

2. 

3. 

**Top Down**

1. 

2. 

3.
Designing in Place

• Also known as “Designing in Context.”
• Solid models are created so that the part and assembly coordinate systems line up in the assembly.
• Neighboring parts can be used as reference.
Bottom-up Modeling Limitations

• Making a design change may require manually changing many interacting parts.

• Geometry and parameters can be referenced between parts, but:
  – It is often done in an ad-hoc, untracked way
  – Parts may change unexpectedly, in unexpected ways.
  – Circular dependencies can be created.

• Top-down modeling can be done in a way that avoids these problems.
Top-down Modeling using Control Models

• We will use the Control Structure concept to do systematic, parametric top-down modeling.

• The key to the Control Structure concept is that one or more control models are used to control the actual parts in the assembly.
  – A control model is not an actual part that is manufactured.
  – It is only a conceptual part, containing a sketch or general shape of the overall design and important design parameters.
  – It acts as a template or skeleton for the entire design.

• This design concept relies heavily on inter-part geometry references and inter-part equations.
Inter-part Geometry References

To create an inter-part geometry reference in SolidWorks:

1. Open the assembly containing the source and destination parts.
2. Hide all components except the source and destination parts.
3. Make the destination part to be the work part. (Right-click on the destination part and select “Edit Part” ( ).)
4. Identify and use modeling operations that can replicate the source geometry. E.g.:
   - Copy a reference plane by creating a reference plane coincident with it.
   - Copy sketch curves by creating a sketch and using “Convert Entities” ( ).
   - Copy faces by using an offset surface operation, with an offset distance of ‘0’.
   - Copy a solid body with ‘0’ translations in each direction.
Inter-part Geometry References

• An alternative way of creating an inter-part geometry reference in SolidWorks is to:
  1. Open the destination file by itself.
  2. Select “Insert” → “Part…”
  3. Select the type of objects to copy (“transfer”). All instances of this type of object will copied.
  4. Click on “OK” without clicking in the graphics window. (Clicking in the graphics window will put the copy in the wrong place.)

• A third alternative (for solids only) is to:
  1. Open the source file by itself.
  2. Use solid modeling operations to create a single solid for each part.
  3. Select “Insert” → “Features” → “Save Bodies…”.
Inter-part Geometry References

• Changing & Exchanging Files
  – Inter-part geometry references keep track of the file name and the feature for the original. Changing either of these outside of SolidWorks will cause the link to become broken and the copied feature will not update properly.
  – To change the name of a source file:
    1. Open the destination file, the assembly file, and the source file
    2. Use “File” → “Save As…” to save the source file to a new part name.
    3. Resave the destination and assembly files.
  – To change a feature in a source file:
    1. Open the source file, make the change, and save the file.
    2. If they are not already open, open the destination and assembly files.
    3. Resave the destination and assembly files.

• Note that feature changes that cause geometry (faces, edges, etc.) to be created or destroyed will result in problems that need to be rectified in the destination file the same as if the feature dependency occurred in the same file (Persistent Naming problem).
Inter-part Geometry References

- Identifying Broken Links
  1. Open the destination part and look at the Feature Manager.
  2. If a feature has “->”, right-click on the feature and select “List External Refs…” to see which file is the source file.
  3. If a feature has “-> ?”, the source file needs to be opened.
  4. If a feature and its part display a warning (⚠️) the link needs to be repaired.

- Fixing Broken Links – source file name changed
  1. Open the assembly file and select the correct source file when prompted.
  2. Open the destination file.
  3. Save the assembly and destination files.

- Fixing Broken Links – source geometry changed
  1. Open destination part in context of assembly, with source part also showing.
  2. Edit the feature showing the warning.
  3. Reselect the referenced geometry.
Inter-part Equations

• Variables in the Equations table can be expressed in terms of variables from other part files.

• To use inter-part equations in SolidWorks:
  1. In the “Value/Equation” column of the “Equations, Global Variables, and Dimensions” window for the destination file, reference the variable from the source file using syntax such as "myvariable@source<1>.Part@Assem.Assembly” (I have not gotten this to work myself.)
  2. Alternatively, in the “Equations, Global Variables, and Dimensions” windows:
     1. For the source file, check “Link to external file:” and select “Create new file” in the “Link Equations” window and click on “Link” (and then “OK”).
     2. For the destination file, check “Link to external file:” and select the text file created from the source file.
     3. Update the text file manually, then click on “Import…” to update values in both the source and destination files.
Top-Down Modeling Procedure

1. Pick a “Team Leader.”
   Team leader does Steps 2-7 with help from team.

2. Create a “control part.”
   Use the term “master” or “control” in the name of the part.
   E.g., “master_layout”, “shape_master”, or “control_sketch”.
   It must contain all important design geometry and variables.

3. Create an “assembly file.”

4. Insert “control part” into “assembly file.”

5. Create empty components.
   Create empty part files (1 thru n) and add to “assembly file.”
   These may represent individual parts or sub-assemblies.
Top-Down Modeling Procedure

• In the Assembly Manager, the parts show up as:

- Assembly part file
- Component parts
- Control part
Top-Down Modeling Procedure

6. Copy geometry from the “control part” to “Parts 1 thru n.”
   Use inter-part geometry references and inter-part equations to copy geometry and variables from the control part to component parts, as required.

7. “Team leader” gives “Parts 2 thru n” to “Person 2, Person 3, … Person n.”

8. “Team leader” models “Part 1,”
   “Person 2” models “Part 2,”
   “Person 3” models “Part 3,”
   …
Top-Down Modeling Procedure

Team leader

Computer 1

Person 2

Computer 2

Person 3

Computer 3

“Persons 2 thru n” should not have the “Assembly file” or the “Control part”!!!
Top-Down Modeling Procedure

9. When “Parts 1 thru n” are fully modeled:
   1) “Person 2 thru n” give “Parts 2 thru n” to “Team leader.”
   2) “Team leader” copies files from “Person 2 thru n” to folder with assembly. Operating system will ask if you want to replace original files. Say “Yes.”

10. Open “Assembly File” and check that it is correct.
Top-down Modeling Rules

1. What geometry should be in the Control Part?
   – If you were to create a sketch of your design on paper, what would be in that sketch? Probably, whatever is in that sketch should be in the Control Part.
   – It should include geometry and parameters for the whole, overall design.
   – The Control Part should contain geometry and parameters that are important to more than one part.
Top-down Modeling Rules

2. Top-down and bottom-up techniques can be used in the same product design.

3. These parts SHOULD be controlled from the Control Part:
   - part contains geometry that is related to geometry in another part (E.g., automobile door and fender shape must match a master shape.)

4. These parts SHOULD NOT be controlled from the Control Part:
   - part is “off-the-shelf”
   - part appears in other assemblies
   - part appears in other products
Top-down Modeling Rules

5. **ALL** inter-part geometry references and inter-part equations **must be** directed from the Control Part to the other part.

6. Parts that are created from the Control Part **usually** should not be constrained in the assembly. (They do not need to be, since they are designed in place.)