



FeatureManager Design Tree Conventions

This article describes the issue that can cause a drawing to include a flattened drawing view, where the sheet metal part is not flat.


The FeatureManager design tree on the left side of the SOLIDWORKS window provides an outline view of the active part, assembly, or drawing.


The FeatureManager design tree uses the following conventions:

- A  symbol to the left of an item's icon indicates that it contains associated items, such as sketches. Click  to expand the item and display its contents.

To collapse all expanded items at once, press Shift+C or right-click the document name at the top of the tree and select Collapse Items.



- Sketches are preceded by


 over defined

 under defined

 the sketch could not be solved


No prefix fully defined

- Features, parts, and assemblies are preceded by the rebuild symbol  if a change has been made that requires the rebuild of the model.
- Parts are followed by the lock icon  if they have been frozen by the freeze bar.
- Errors and warnings are displayed next to part, feature, or sketch icons and described in tooltips (when the pointer hovers over the item) and in What's Wrong?

 an error in the model

 an error with the feature

 a warning underneath the node

 a warning with the feature

- Positions of assembly components are indicated by:

(+) over defined

(-) under defined

(?) not solved

(f) fixed (locked in place)

- In an assembly, each instance of the component is followed by a number in angle brackets $\langle n \rangle$ that increments with each occurrence.
- Assembly mates are preceded by:

(+) involved in over defining the position of components in the assembly

(?) not solved

- The state of external references is displayed as follows:

-> If a part or feature has an external reference, its name is followed by **->**. The name of any feature with external references is also followed by **->**.

->? If an external reference is currently out of context, the feature name and the part name are followed by **->?**.

->* The suffix **->*** means that the reference is locked.

->x The suffix **->x** means that the reference is broken.

You can hide the x. Click Tools > Options > System Options > External References and clear Show "x" in feature tree for broken external references.

- While a drawing view updates, its icon in the FeatureManager design tree changes to: .