

EDU[**CAD**]TION **STUDY GUIDE**

# **How to Repair Broken References in CAD Assemblies**

Chapter 2, Lesson 7

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### Understanding Parametric Design

- Parametric modeling allows changes made to one feature (like dimensions or sketches) to automatically update all related features.
- This means you can modify your design without rebuilding it from the ground up. In this example, the Arduino board model is linked to the main assembly, allowing updates to transfer between files.

## Making Changes to Linked Components

- Open your Arduino file and locate the base sketch that defines its shape. By right-clicking and selecting Edit Sketch, you can modify the original dimensions.
- For example, changing the board's length from 2.7 inches to 5 inches immediately updates the mounting holes and overall geometry. Once finished, click '**Finish Sketch**' and '**Click Save**'.
- Back in the main assembly file, the first thing you'll notice is none of those changes came though. You'll see a '**warning icon**' in the upper left corner indicating that a linked component is out of date (see Figure 1.1).
- Click the '**warning icon**' to refresh the model. The entire enclosure will automatically adjust to the new 5-inch dimension, including the box, lid, and mounting pads. This demonstrates how parametric references maintain design consistency (see Figure 1.2).

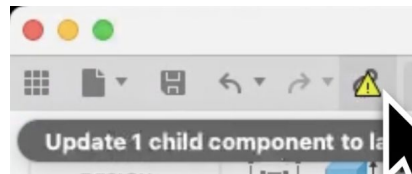


Figure 1.1

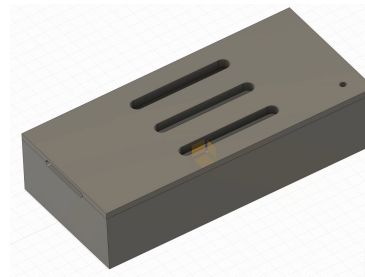


Figure 1.2

### Identifying Lost References

- Sometimes, design changes can break references between parts. When a reference is lost, you'll see '**yellow warning icons**' in the timeline.
- These indicate that Fusion can't find the original feature or sketch it was referencing.
- For example, if the reset button hole is deleted or moved, any projections or extrusions based on it will show a warning. If you go to your timeline, you'll see a yellow icon (see Figure 1.3).

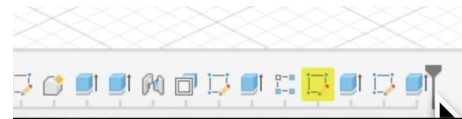


Figure 1.3

### Fixing Lost Sketch References

- To fix missing references, right-click on the affected sketch (**yellow icon**) and select '**Edit Sketch**'.
- If the projected geometry no longer exists, delete it, then use **Create > Project** to re-project the new geometry.
- Click **OK** to reapply the projection, restoring your sketch relationships. Always check for additional yellow icons that may have propagated to later features

### Repairing Extrusion References

- After restoring your sketch, you may notice the Extrude feature now shows an error.
- Right-click the affected **Extrude** in the timeline and select **Edit Feature**. It will show it's '**missing profiles**' because we just erased it (see Figure 1.4).
- To fix, under the **Profiles** list, re-select the new sketch area that defines the feature, then click OK. The feature will regenerate correctly using the updated geometry.

**Note:** When repairing multiple broken references, always start from the earliest error in the timeline (the farthest left yellow icon)

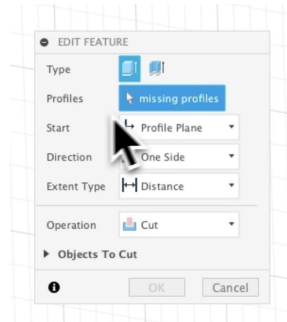


Figure 1.4

### Summary

By leveraging parametric design and understanding how to repair lost references, you can keep complex assemblies flexible and stable. Small changes no longer require full redesigns, instead, you can update individual parameters and rely on your references to maintain alignment and accuracy.

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