Board-level Troubleshooting

TIPL 1901
TI Precision Labs – Op Amps

Presented by: Joseph Serritella

Prepared by: Joseph Serritella, Ian Williams, Tim Green, Paul Rheinheimer

Introduction:

Hello, and welcome to TI Precision Labs discussing board level troubleshooting. Board level troubleshooting is the first in a series of guides, which will provide best practice application debugging techniques; known as the quality and application handbook. Various engineering checks are outlined to help determine the root cause of an issue. The first training focuses on a technique known as the A-B-A swap, in-addition an introduction of PCB parasitics and the importance of PCB cleanliness are reviewed.

Important Note

Application handbook and troubling shooting guide:

- The troubleshooting portion of Precision Labs presents a number of best practices to assist the customer with application debugging.
- Various engineering topics and techniques will be presented throughout the series:
  - Board level trouble shooting: ABA Swap
  - Parasitics
  - Board cleaning
  - Curve trace analysis
  - Digital Multi Meter (DMM)
  - Precision measurements
  - Power amplifier best practices
  - High speed amplifier best practices
  - Design for stability
  - Best practice PCB Designs

Important note:

As discussed, the application handbook is a section of TI Precision Labs dedicated to troubleshooting and debugging. Various engineering checks will be presented including the following topics but not limited to: A-B-A swap, PCB parasitics, PCB cleaning techniques, curve trace analysis, digital multi meter use, precision measurement techniques, power amplifiers best practices, high speed amplifiers best practices, stability and PCB design best practices. The examples presented can be applied to applications containing integrated circuits of varying complexity.

Information discussed and presented within the troubleshooting guides is provided “AS-IS”. The content is provided with the intent to assist in debugging and analysis processes.

A-B-A Swap

Overview:

- **A-B-A swap** is a verification technique that is crucial during the application troubleshooting process
- Provides confirmation that non-conforming behavior follows the **D**evice **U**nder **T**est (**DUT**)
- Completion of an A-B-A swap is required prior to returning a DUT to TI devices for analysis

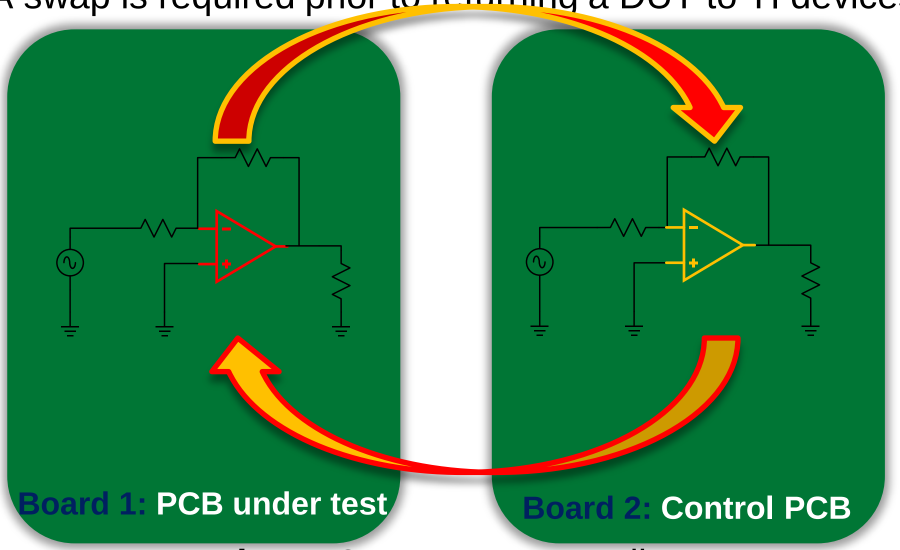


Figure 2-1: A-B-A swap diagram

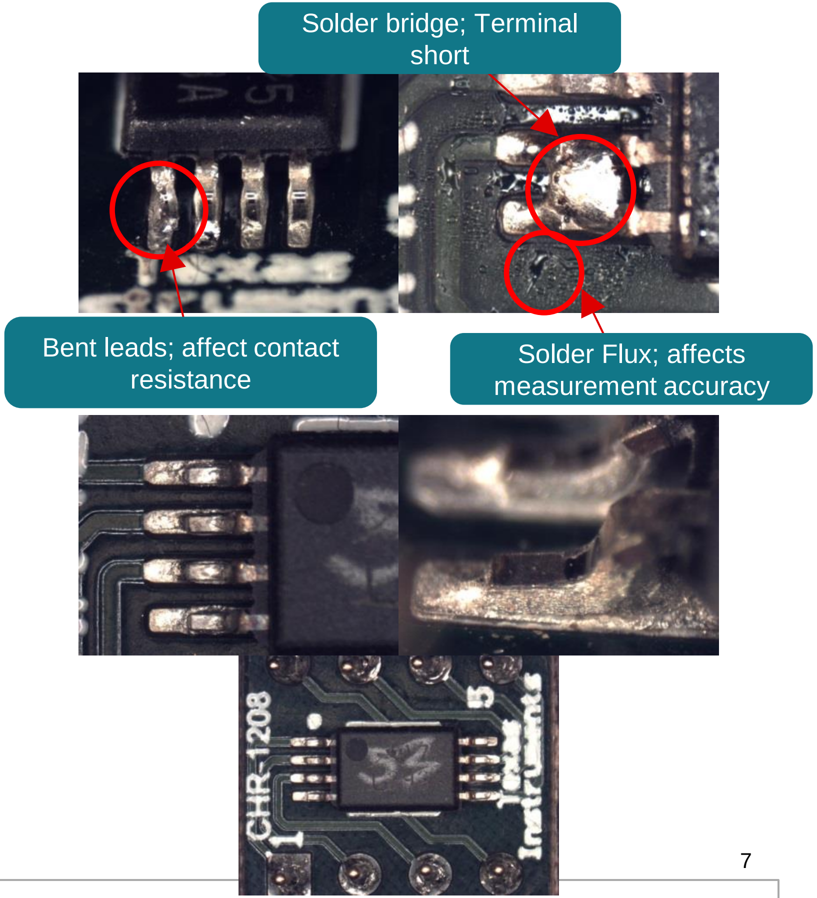
A-B-A Swap:

The A-B-A swap is a critical troubleshooting technique highly recommended during the debugging process. The A-B-A swap helps characterize how the application performs using various devices of the same model. In-addition, the A-B-A swap technique verifies if the application failure characteristics follow the device under test. Device under test will be referred to as the DUT for the remainder of the presentation. Completion of an A-B-A swap is strongly recommended before requesting verification analysis through the authorized TI distributor or TI.com. Figure 2-1 provides an animation of the A-B-A swap to depict the process. The DUT is interchanged with a matching device from a known good PCB. Additional details will be provided within the remaining slides.

Optical Inspection

<u>Visual inspections:</u>

- A thorough visual inspection should be completed prior to the initiation of an A-B-A swap
- Visual inspection improves the detection rate of inconspicuous (hidden) issues which may be overlooked:
  - Utilization of a microscope is highly recommended
    - Bent leads/lifted leads
    - Solder bridges
    - Flux contamination
    - Damaged traces
    - IC orientation
- Visual inspection can save valuable time and effort
- Key inspection areas are:
    - Leads/footprints
    - Surrounding circuitry
    - Connected traces



Optical Inspection:

A troubleshooting technique often overlooked during the debugging process is completion of a thorough optical inspection. Automated inspections are generally completed during the application manufacturing process however; some PCB issues may go unnoticed by the automated tool. Low tech assembly issues can easily be missed without a visual inspection; resulting in countless hours spent electrically troubleshooting an application. Therefore, a manual inspection should always be completed before beginning the A-B-A swap or another analysis technique. Utilization of a microscope is highly recommended for the optical inspection process.

Examples of frequent assembly issues are: bent or lifted leads, solder bridges, flux contamination (more on flux later), damaged PCB traces and IC orientation. Visualizations of typical assembly issues are highlighted in the top right figures. Including bent leads which affect contact resistance. Solder bridges which result in terminal shorts. And Flux which can affect measurement accuracy. A visual inspections is an early detection technique which can save valuable time and engineering resources. Critical inspection areas are: leads and footprints, surrounding circuitry, and immediate PCB traces. In addition, it is always beneficial to obtain time zero reference pictures of the PCB for records.

Benefits of A-B-A

Why complete an A-B-A swap:

- The A-B-A technique applies to troubleshooting both complex and simplistic electronic applications
  - Valuable analysis data is obtained from an A-B-A

- An A-B-A swap can identify:
  - Confirmation that the failure mode follows the device under test
  - Helps eliminate the possibility that the issue is caused by an interaction with another part on the board
  - Rules out PCB issues caused by the assembly process

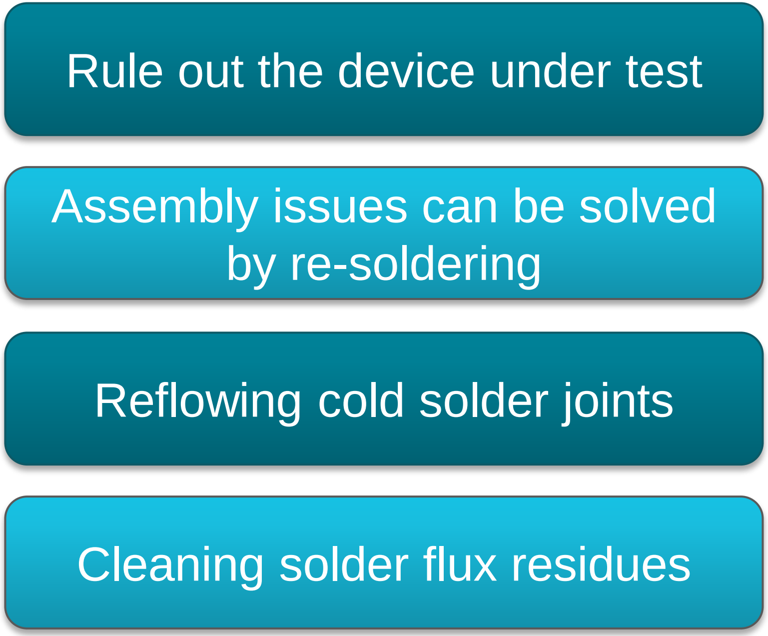


Figure 5-1: Key benefits of the A-B-A swap

Benefits of A-B-A:

In summary the A-B-A swap is an extremely powerful troubleshooting technique that applies to both complex and simplistic application circuits. In addition to verifying the electrical characteristics of the DUT in various PCBs, the A-B-A swap helps to confirm that the problem follows, or does not follow, the device under test.

The A-B-A helps eliminate the possibility that the issue is caused by an interaction between the DUT and other electrical components. In addition, issues with assembly can also be identified through the completion of an A-B-A swap. Re-soldering a component may solve the problem. Typical assembly issues which could affect application performance are: Solder bridging, cold solder joints and flux contamination. Reflowing or cleaning can result in the application functioning as expected, allowing for the issue to be closed before further analysis is performed.

A-B-A Process

A. Remove the suspect TI part (A) from the original failing board.

B. Replace the suspect TI part (A) with a known good TI part (B) and check if the original failing board is now working correctly.

A. Mount the suspect TI part (A) to a known good board and see if the issue is observed

- The last step is critical to exclude the possibility that the issue is caused by an interaction with another part on the board.
- It is recommended to re-solder part (A) to the original PCB to confirm if the original setup passes or does not function properly:
  - Cold solder joint, flux contamination, etc..

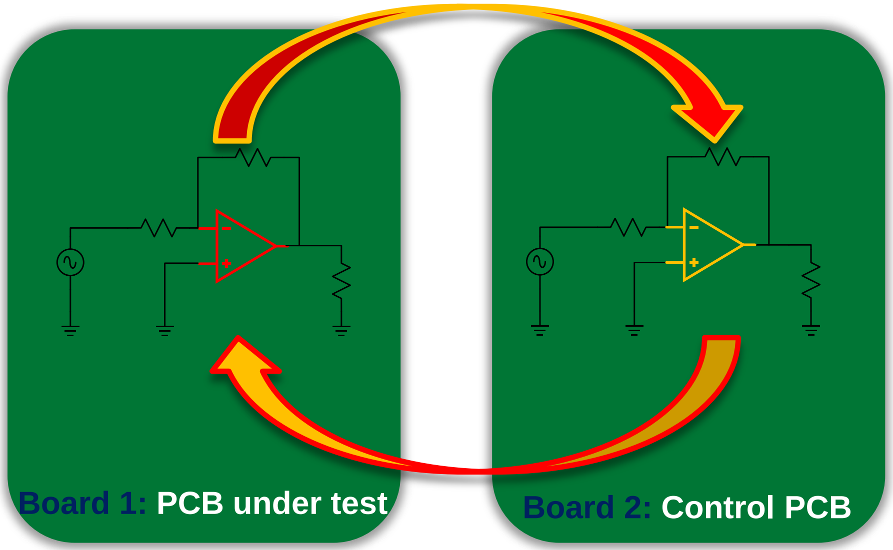


Figure 6-1: A-B-A swap diagram: Unit A | Unit B. Units must be carefully de-soldered and re-soldered. Take care to prevent thermal damage to the units.

A-B-A process:

Slide 6 provides a detailed explanation of the A-B-A swap methodology. The process includes 2 printed circuit boards and two integrated circuits of interest; the device under test and a control unit. The first step of the process is to record the electrical characteristics of both systems. Recording output voltage or voltage offset of a system is a good example. An inverting circuit is depicted in figure 6-1. Once the data is successfully recorded, the A-B-A swap process can begin. Carefully de-solder and remove the suspect device (A) from the non-conforming application PCB. Device (B) should be removed from the control PCB as well. Replace the suspect TI part (A) with a known good TI part (B) and check if the original failing board is now working correctly. Be sure to record the data for future records. Mount the suspect TI part (A) to a known good board and see if the issue is observed again. Finally, the last step will be to return device A and B to the original PCBs the units were mounted on. This final step is to confirm if the original electrical results of device A and the PCB under test remain unchanged. If device A and the PCB under test now function as expected the A-B-A swap likely fixed an assembly defect such as flux contamination or cold solder joints. Remember, it is extremely important to ensure care is taken during the de-soldering and re-soldering process.

De-soldering and Cleaning

De-soldering:

- The plastic mold compound of the part package is soaking humidity to some extent.
- Therefore, all TI parts, including MSL1 classified TI parts, must be dry-baked according to IPC/JEDEC J-STD-033 before de-soldering
- TI recommends use of a rework station allowing control of the soldering temperature according to the JEDEC soldering profile
  - **Details found at: http://focus.ti.com/lit/an/slva439a/slva439a.pdf**

Cleaning:

- Precision electrical measurements may be dependent on the cleanliness of the PCB
- An optical inspection should be completed after the de-soldering process and A-B-A swap.
  - Excess solder and flux should be removed promptly

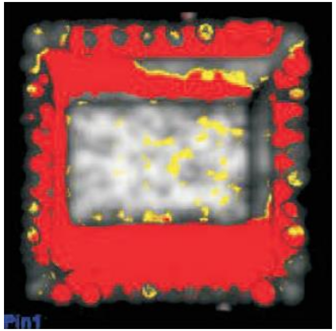


Figure 7-1: A failure analysis technique know as CSAM (Scanning Acoustic Microscopy). CSAM is used to determined if delamination exist. Example of component delamination after removal from PCB is shown in figure 7-1. The delamination is an example of “popcorn” delam. Popcorn delamination commonly occurs due to rapid evaporation of moisture within the package.

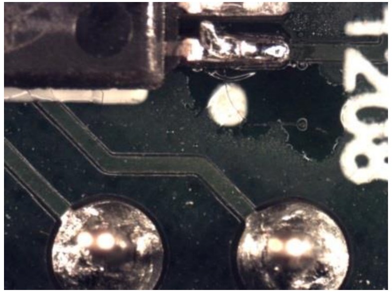


Figure 7-2: excess solder remaining on the PCB. Prior to electrical analysis remaining solder flux should be removed from the board.

De-soldering and Cleaning:

Proper soldering and removal of the component from the PCB is critical to the A-B-A swap process and for failure analysis submissions. Patience and due diligence is required to safely remove components from the non-conforming PCB. For example, integrated circuits which utilize plastic mold compound, encapsulate, may absorb humidity to some extent. Therefore, TI parts should receive a dry-bake step, per to IPC/JEDEC J-STD-033, before de-soldering the component.

Figure 7-1 is an example of the de-soldering process resulting in delamination of the component. Delamination can be explained as the separation of two materials at an interface. For example, the separation of the mold compound, encapsulate, to the leadframe of the component. In the case of figure 7-1 the delamination can be referred to as “popcorn” delamination. Popcorn delamination commonly occurs due to the rapid evaporation of moisture from within the package. Skipping the dry bake step prior to de-soldering may result in this anomaly. The area highlighted in red represents delamination caused by the rapid expansion and evaporation of moisture which seperated the mold compound from the leadframe. This could result in lifted bond wires or an impact to electrical parameters.

Examples and recommendations of soldering controls can be found on TI.com. Details can be found using the link provided in this training.

In addition, board cleanliness should be monitored during the soldering and de-soldering process. Precision measurements may be affected by excess flux or board contamination leading to skewed measurements. For example, excess solder flux, shown in figure 7-2, may impact electrical characteristics of the application resulting in an incorrect reading from an analog to digital converter. Prior to electrical analysis excess solder flux should be removed from the PCB. Therefore, an optical inspection should always be complete prior to, and after, soldering a component.

Solder Flux

<u>What is solder flux:</u>

- A chemical agent used to facilitate the soldering of components to a printed circuit board
- Solder flux serves three main purposes:
  - Removes oxidation from surfaces to be soldered
  - Seals out air, preventing further oxidation
  - Improves “wetting” characteristic of the liquid solder
    - Solder flows more easily onto solder pads and device pins
- Many different types of solder flux
  - Resin, organic, inorganic
  - Liquid, solid, paste

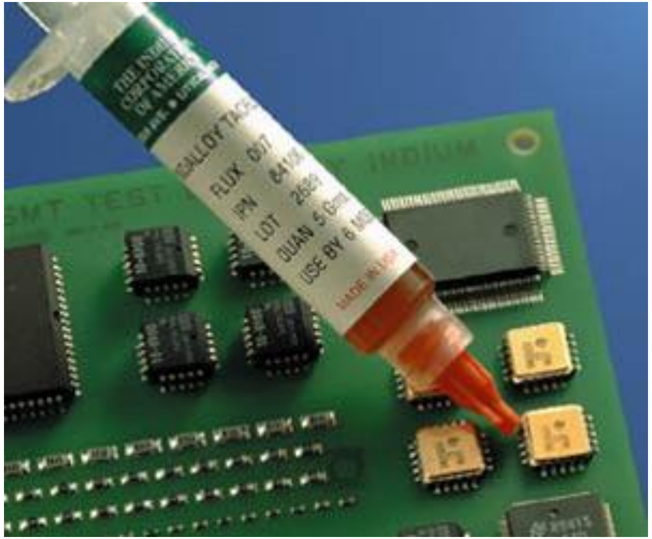


Figure: Example image of flux commonly used in the soldering of electronic components to PCB boards.

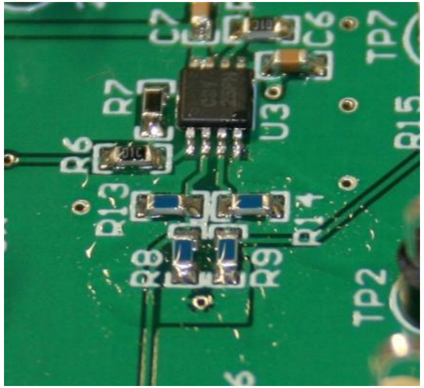
Solder Flux:

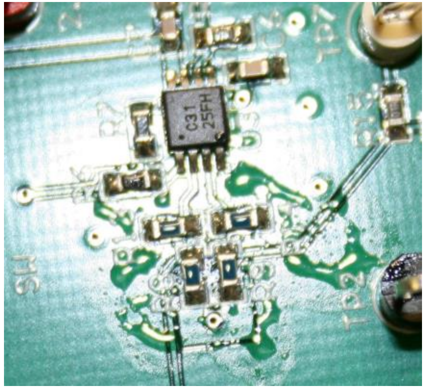
As discussed in the previous slides, electrical measurements can be impacted by the cleanliness of the printed circuit board. Therefore, it is important to understand common processes that can relate to an increase in PCB parasitics. Solder flux is commonly used during the PCB assembly process and board prototyping. Recall that flux is a chemical agent used to facilitate soldering of components onto a PCB. Three of the main purposes are: Oxidation removal from the surface of the PCB, seals out air, preventing oxidation, and improving the “wetting” characteristics of the liquid solder. It is important to remember that solder flux comes in many different forms such as: resin, organic compounds and inorganic compounds. Also, solder flux can be found in either a liquid, solid or paste form. It is important to remember to remove excess flux no matter the type of solder.

Solder Flux Contamination

Impact of flux contamination:

- **Deterioration of surface insulation resistance!**
  - Affects electrical measurement accuracy
  - Introduction of parasitic electrical paths
- **Contamination of sensitive parts: Connector contacts, mechanical switches, MEMS assemblies**
  - Attributes to early break down of mechanical components
- **Growth of whiskers between nearby traces**
  - Accelerate electrical breakdown of traces. Resulting in potential damage to circuity





Solder Flux Contamination:

How can flux contamination result in erroneous measurements? The first example is related to insulation resistance. Excess flux impacts the overall impedance between electrical connections on the PCB. Shifts in impedance will affect electrical measurement accuracy through the introduction of parasitics electrical paths on the PCB. An example of parasitics will be shown on the following slide.

Flux may contaminate sensitive components like switches, mechanical Micro-Electro-Mechanical Systems (MEMS) and contactors. Over long term operation mechanical components may breakdown more quickly.

Lastly, whiskers may form between traces resulting in potential shorts or unwanted electrical paths. In some cases whisker may lead to damage to the integrated circuit and or PCB.

Cleaning Solder Flux

Cleaning methodology:

- Solvents such as **acetone** and/or **flux remover** can be utilized to limit the impact of flux contamination
  - These solvents may be utilize to remove conformal coatings which are soluble.
- An **ultrasonic bath** can be utilized to improve the effectiveness of the cleaning process
- Unfortunately all coatings my not be soluble in acetone or by using a ultrasonic bath with acetone.
  - Additional methods may be required to remove coatings to ensure the DUT is in a testable condition.

Figure 12-1: examples of acetone and flux remover which can be used to remove flux

Figure: 12-2 An example sonic bath tool, Branson 3510

Important: ensure all components are safe for use with the cleaning solvents and methods selected for use

.

Cleaning Solder Flux:

There are a number of cleaning methodologies an engineer can apply to the printed circuit board to reduce the effects of flux contamination. A large portion of these methods include a solvent such as Acetone and flux remover. Ensure all components are safe for use with the cleaning solvents before attempting to clean a PCB. Conformal coatings and flux are generally soluble when exposed solvents. Some visual examples are shown in figure 12-1.

To improve the overall effectives of PCB cleaning, an ultra-sonic bath can be utilized. An example of a tool used in a prototyping environment is the Branson 3510. Lastly, all coatings may not be soluble in acetone or by using a ultrasonic bath with acetone therefore additional methods may be required to remove coatings to ensure the DUT is in a testable condition. Please refer to the flux supplier’s technical documents for details.

Printed Circuit Board Parasitics

Effects of parasitics:

- An ideal printed circuit board has no impact on a circuit's performance
- In reality, certain PCB layout techniques, electronic components, and other materials create "extra" circuit elements; known as **parasitics**!
  – Examples: thermocouples, resistors, inductors, capacitors, **cold solder joints** and **contamination**
- Assembly issues such as flux contamination and cold solder joints can be resolved through the **A-B-A swap** process

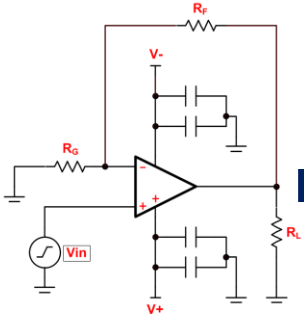


Figure 10-1 : op-amp circuit as designed and simulated

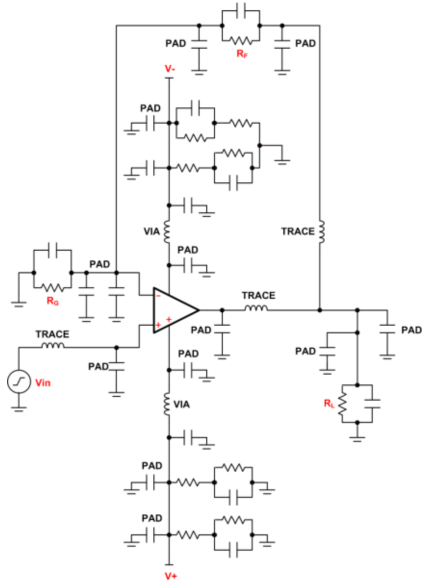


Figure 10-2: op-amp circuit including parasitics. Board contamination could further increase the number of parasitic paths

Source: "A Practical Guide to High-Speed Printed-Circuit-Board Layout"

Printed Circuit Board Parasitics:

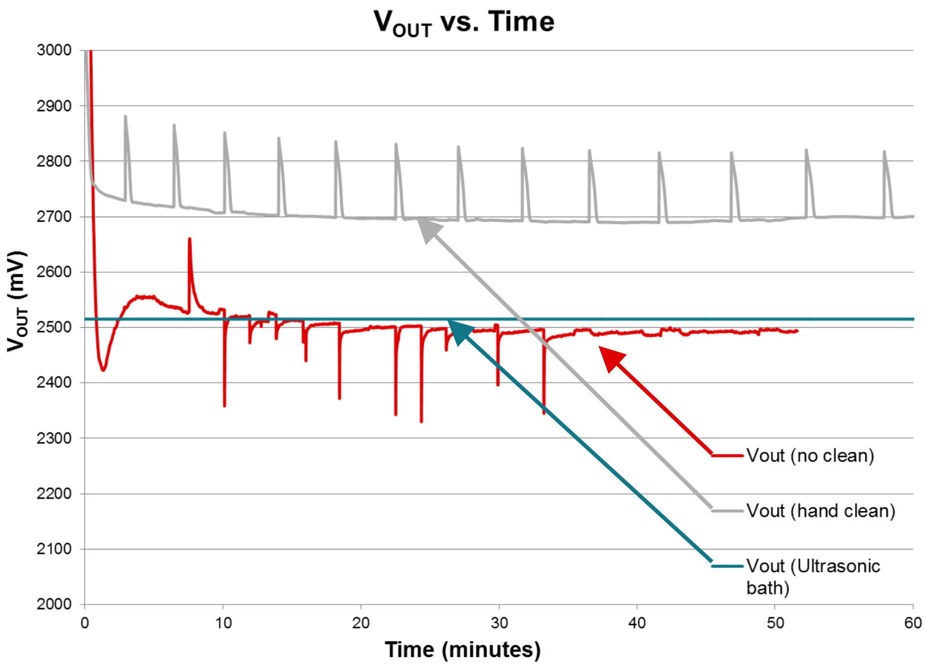
Why is removing flux important? Consider parasitic connections and their effects on an application from a schematic perspective. Ideally, a printed circuit board and contaminates have no impact on an application's performance. For example, figure 10-1 depicts a schematic of an inverting amplifier as originally designed and simulated by an engineering team. However, in reality, electronic components, and other materials create "extra" circuit elements; known as parasitics.

Let's review figure 10-2 however, this time please consider how the schematic would be represented with additional parasitics for each component. As shown in the real world schematic, the schematic routing has change greatly with the addition of the parasitic components. The values of the parasitics can vary greatly depending on the cause. Some common examples are thermocouples, resistors, inductors, capacitors. Don't forget parasitics caused by PCB assembly and soldering issues! Cold solder joints and flux contamination can also add to the overall error of the system.

Effects of Solder Flux on Circuit Measurements

Impact of flux contamination and assembly issues:

- Excess solder flux on PCBs can cause DC voltage **errors!**
  - These errors can be random in nature and unpredictable
  - Assembly defects such as "cold solder joints" can have similar parasitic effects
- Proper cleaning of Flux can limit the occurrence of production issues
- PCBs should receive a thorough cleaning during the A-B-A swap process if excess flux is detected.



Effects of Solder Flux on Circuit Measurements:

We have seen an example how parasitics can be represented from a schematic perspective. It is also valuable to see real world effects of parasitic components, such as flux contamination, on an application circuit. Parasitics caused by flux contamination can attribute to the overall DC error of the circuit. Even worse, since flux contamination is non-ideal the effects can be random and unpredictable. The graph shown on the right represents the electrical measurements over time of three examples; Uncleaned system, hand cleaned and cleaning using an ultra-sonic bath. The output voltage of the system was recorded for 60 minutes in all three cases. It is critical to note the output voltage over time for each example. Red represents the board when no cleaning is applied, grey represents hand cleaning and blue used an ultra-sonic bath to clean the PCB. The output voltage appeared unpredictable during the measurement process when no cleaning was applied to the board. A DC offset and a periodic spike was seen when only a hand cleaning method was used to remove excess contaminates. However, use of an ultra-sonic bath to remove excess flux resulted in a smooth signal centered at approximately 2.5V.

From the graph it is easy to see how proper cleaning can limit the occurrence of production issues caused by contamination such as flux or assembly issues such as cold solder joints.

PCBs should receive a thorough cleaning during the A-B-A swap process if excess flux is detected.

Thank You!
Please try the quiz to test your knowledge