

# **PSpice Known Problems and Solutions**

**Product Version 17.4-2020  
June 2020**

© 2020 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Product PSpice contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. vtkQt, © 2000-2005, Matthias Koenig. All rights reserved.

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

All other trademarks are the property of their respective holders.

**Restricted Permission:** This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

**Restricted Rights:** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

---

# Contents

---

<u>Known Problems and Solutions in PSpice</u> .....	5
<u>CCR 01721596: In the new Simulation Settings dialog, loading a simulation profile with spaces in its name after editing causes an issue</u> .....	5
<u>CCR 1566898: Save data on markers in not working -no .probe statement</u> .....	5
<u>CCR 19462: Cannot use relative tolerances on parameters used for DEV in Monte Carlo analysis</u> .....	6
<u>CCR 22481: Part names with spaces in the name result in "Error in Open Alias" when simulating</u> .....	6
<u>CCR 152246: Error in displaying plot, if the simulation profile name includes any special character.</u> .....	6
<u>Known Problems and Solutions in Capture-PSpice Flow</u> .....	8
<u>CCR 332772: Issue with the Markers flow in hierarchical designs in OrCAD Capture</u>	8

# PSpice Known Problems and Solutions

---

---

# PSpice Known Problems and Solutions

---

This Known Problems and Solutions document describes important Cadence Change Requests (CCRs) for PSpice and PSpice Simulator<sup>1</sup> and tells you how to solve or work around these problems. For information about CCRs that are fixed for this release, see PSpice Product Notes.

**Important:** Only known problems and solutions available at release time are available in this document.

## Known Problems and Solutions in PSpice

This section lists the known problems in PSpice and tells you how to solve or work around these problems.

### **CCR 01721596: In the new Simulation Settings dialog, loading a simulation profile with spaces in its name after editing causes an issue**

**Description:** In the new Simulation Settings dialog, when you edit and save a simulation profile with a space character in its name or design path, the design gets corrupted.

**Solution:** Do one of the following to resolve the issue:

- Add an environment variable, `PSPICE_SIMSETUP_1722`, and launch the legacy Simulation Settings dialog.

The environment variable can have any value.

- Remove all space characters from the profile name or design path.

### **CCR 1566898: Save data on markers in not working -no .probe statement**

**Description:** In the Capture-PSpice flow, if `At Markers Only` option is selected for any of the data collection options in the Data Collection tab of the Simulation Settings window, the

---

1. Depending on the license and installation, either PSpice or PSpice Simulator is installed.

## PSpice Known Problems and Solutions

### PSpice Known Problems and Solutions

---

trace related to markers is not added in the Simulation Results window, nor is an entry related to markers is added in the circuit file. For example, if a voltage marker is added to the RESET netname, and the At Markers Only option is selected for the voltage option, there will be no trace related to the voltage marker in the simulation window and the circuit file.

**Solution:** To add a trace in PSpice in Capture - PSpice flow for the At Markers Only option, either add a PSpice directive in a Capture schematic, or add an include file (.inc) in PSpice.

To add a PSpice directive in Capture, select *Place – Text*. Add @PSpice: keyword in the first line before adding the marker-related text, such as `.PROBE64 N([Netname])`.

To add an include file in PSpice, add the file to the design from *Simulation Settings window – Configuration Files tab – Include Category*. Click *Add to Design* and *OK*.

### CCR 19462: Cannot use relative tolerances on parameters used for DEV in Monte Carlo analysis

**Description:** When running a Monte Carlo (.MC) analysis, a DEV tolerance that is a parameter (for example, DEV={ATOL}) will work only if it is an absolute tolerance.

**Solution:** If using Capture, Design Entry HDL, or Schematics for design entry, you can create an expression for DEV that multiplies that value by the relative tolerance divided by 100. For example, given a relative tolerance RTOL%, if the property being tolerated is VALUE, set DEV={VALUE\*RTOL/100}.

### CCR 22481: Part names with spaces in the name result in “Error in Open Alias” when simulating

**Description:** If a part reference contains a space (e.g., MY PART), then the simulation fails to run. The PSpice window shows the error: “Error in Open Alias.”

**Solution:** Use an underscore (\_) instead of a space in the part reference.

### CCR 152246: Error in displaying plot, if the simulation profile name includes any special character.

**Description:** If the simulation profile name includes a special character, an error is encountered while displaying plot. For example, if the simulation profile name is trans`.sim and you are trying to plot measurement vs. measurement, the following error message appears:

## PSpice Known Problems and Solutions

### PSpice Known Problems and Solutions

---

“One or more required header items are missing from csdf file.  
Error trying reading data file.  
The data file is empty.  
Analysis failed.”

**Solution:** Avoid special characters in the simulation profile, design, and schematic names.

## **Known Problems and Solutions in Capture-PSpice Flow**

This section lists important Cadence Change Requests (CCRs) when using Capture with PSpice and tells you how to solve or work around these problems.

### **CCR 332772: Issue with the Markers flow in hierarchical designs in OrCAD Capture**

**Description:** When you have two or more current markers in a design and delete the markers from the Probe window, you can enable only one marker in OrCAD Capture.

**Solution:**

To solve the issue, do the following:

1. In CAPTURE, choose *PSpice – Markers – List*. De-select and then select the markers to enable all the markers.
2. Select any one of the markers and rotate it. Now you are able to select the other marker as well.
3. Select markers on one of the descend down schematic page to enable markers on this page, close down the other schematic, and reopen it. You are able to place all markers on it.