

SPB 16.3 README for SiP and IC Packaging

CADENCE SPB RELEASE 16.3 README

System-in-Package (SiP) and Allegro Package Designer Products

This document provides detailed information for the System-in-Package (SiP) and Allegro Package Designer (APD) products. For general release information, refer to the *Cadence SPB Release 16.3 Readme*.

SiP/Allegro Package Designer Product Compatibility Information

The System-in-Package (SiP), Allegro Package Designer (APD), and Cadence 3D Design Viewer products are installed from the Cadence SPB/OrCAD 16.3 Install DVD. The Cadence IO Planner product, used for concurrent co-design, is based on the Encounter Design Implementation (EDI) Systems technology and is installed from the EDI Systems CD ROM image. Both, the Cadence SPB/OrCAD 16.3 Install DVD and the EDI Systems CD ROM images contain compatible versions of the OpenAccess (OA) software.

SiP RF Architect XL provides an integrated environment with the Virtuoso Analog Design Environment (ADE). Virtuoso products are installed from the Virtuoso (IC) CD ROM image.

For SPB 16.3, the compatibility of Cadence SiP and APD products with OA, EDI Systems and Virtuoso is described in the following table.

Product Compatibility Details

Product Name	Version Supported with SPB 16.3
EDI Systems	EDI Systems 09.10
OA	OA22.04.052
Virtuoso (OA database)	IC6.1.3.500.13
Virtuoso (CDBA database)	IC5.1.41 USR6
MMSIM	MMSIM 07.11.x

NOTES:

1. To be able to use the Cadence IO Planner, you must install the EDI Systems CD ROM image.
2. SPB 16.30.000 Base release is validated with a pre-released version of EDI09.10.000. Validation with the EDI09.10.000 Base release will be completed in Q1 2010.

SPB 16.3 README for SiP and IC Packaging

SiP 16.3 Computing Platform Support Matrix

The following table lists the supported operating systems for SiP, APD, and Cadence 3D Design Viewer. For additional information on system requirements, see the *Allegro Platform System Requirements* document.

Computing Platform Support for SiP 16.3 SiP/APD Products

Product name	Win XP Pro	2003 Server	2008 Server	Vista	Solaris 10 sunv4	Linux RHEL 4.0	Linux RHEL 5.0	AIX 5.3
Cadence SiP Digital Architect XL SIP110	Y	Y	Y	Y	Y	Y	Y	N
Cadence SiP Digital Architect GXL SIP125	N	N	N	N	Y	Y	Y	N
Cadence SiP Digital SI XL SIP215	Y	Y	Y	Y	Y	Y	Y	N
Cadence SiP Layout XL SIP225	Y	Y	Y	Y	Y	Y	Y	N
Cadence SiP RF Architect XL SIP410	N	N	N	N	Y	Y	Y	N
Cadence Chip Integration Option SIP625	Y	Y	Y	Y	Y	Y	Y	N
Allegro Package Designer L PX4100	Y	Y	Y	Y	Y	Y	Y	Y
Allegro Package SI L PA7625	Y	Y	Y	Y	Y	Y	N	Y
Cadence 3D Design Viewer PA6605	Y	Y	Y	Y	Y	Y	Y	N

SPB 16.3 README for SiP and IC Packaging

Computing Platform Support for SiP 16.3 SiP/APD Products that are End-of-Sale

Product name	Win XP Pro	2003 Server	2008 Server	Vista	Solaris 10 sunv4	Linux RHEL 4.0	Linux RHEL 5.0	AIX 5.3
Cadence SiP Digital Architect GXL (legacy) SIP120	N	N	N	N	Y	Y	N	N
Cadence SiP Digital SI XL (legacy) SIP210	Y	Y	N	Y	Y	Y	N	N
Cadence SiP Digital Layout GXL (legacy) SIP320	Y	Y	N	Y	Y	Y	N	N
Cadence SiP Digital Layout GXL SIP325	Y	Y	Y	Y	Y	Y	Y	N
Cadence SiP RF Layout GXL (legacy) SIP520	Y	Y	N	Y	Y	Y	N	N
Cadence SiP RF Layout GXL SIP525	Y	Y	Y	Y	Y	Y	Y	N
Allegro IC Package Design Partition Option PA6615	Y	Y	Y	Y	Y	Y	Y	Y
Allegro Package Designer XL (legacy) PA6620	Y	Y	N	Y	Y	Y	N	Y
Allegro Package Designer XL PA6625	Y	Y	Y	Y	Y	Y	Y	Y
Allegro Package SI L (legacy) PA7620	Y	Y	N	Y	Y	Y	Y	Y

NOTES:

- (1) SiP/Packaging products are not supported on the following platforms: Solaris 10 x86, Linux SLES 9, Linux SLES 10, Linux SLES 11, Vista Home Basic.
- (2) Spectre Circuit Simulator is not supported on Windows platforms.
- (3) Cadence IO Planner is not supported on the Windows or AIX platforms.
- (4) There may be additional platform limitations depending on the version of Virtuoso that you are running. Refer to Cadence Online Support for information on known limitations on Virtuoso Computing Platforms.
- (5) Allegro Package SI L (PA7625), Cadence SiP Digital SI XL (SIP215), and Cadence SiP Digital Architect GXL (SIP125) all require a third-party product (PakSi-E) acquired from Apache Design Solutions. Refer to <http://www.apache-da.com/> for platform information.

SPB 16.3 README for SiP and IC Packaging

SiP/Cadence IO Planner/OA Installation and Setup

For SiP concurrent co-design, the SiP Layout, Cadence IO Planner, and OA software must be installed.

The SiP products are installed from the Cadence SPB/OrCAD 16.3 Install DVD. The Cadence IO Planner is part of EDI Systems technology and installed from the EDI Systems CD ROM image. For Cadence IO Planner, install the Encounter Digital Implementation Systems L (EDS100) or Encounter Digital Implementation Systems XL (EDS200) product.

OpenAccess (OA) software is available on both the SPB and EDI Systems CD ROM images. OpenAccess version 22.04.052 is installed under <cadence_install_dir>/oa_v22.04.052 when you install any of the following SPB 16.3 products/options on Linux or Solaris:

- SIP125 Cadence SiP Digital Architect GXL
- SIP325 Cadence SiP Digital Layout GXL
- SiP525 Cadence SiP RF Layout GXL
- SIP625 Cadence Chip Integration Option
- PA6625 Allegro Package Designer XL

Refer to the *OpenAccess Configuration Guide* included in the Cadence product documentation for information on completing the OA installation. This document also describes the additional setup required to run the OA utilities in a standalone mode.

To configure the latest version of OA:

1. Install the EDI Systems software.
(Ensure you install it in a different hierarchy than any other version you have on your system.)
2. Install the SPB software.

NOTE: The order in which you install the EDI Systems or SPB software does not matter.

3. Before running the Cadence IO Planner, set the following:

```
setenv ENCOUNTER <path to your EDI Systems installation>
setenv PATH ${ENCOUNTER}/bin:${PATH}
encounter -version
echo "OA Version is" oaGetVersion
```

SPB 16.3 README for SiP and IC Packaging

SPB and EDI Systems software may contain different embedded builds of OA 22.04. You can see different versions listed by `oaGetVersion` depending on the order in which you have the SPB and EDI Systems bin directories listed in your PATH environment variable. SiP Layout and Cadence I/O Planner will each use its own embedded build of OA and will be able to read/write data when working together in the IC/ Package co-design flow (as they are both based on OA Data Model 4). If you need to use a later version of OA (e.g. for a bug fix) you can do so by setting the `OA_HOME` environment variable before running SiP Layout. For example, if you want to use build OA 22.04.060 installed in `$CDSROOT/oa_v22.04.060`, you will need to set the `OA_HOME` environment variable, as follows.

```
setenv OA_HOME ${CDSROOT}/oa_v22.04.060
```

Using the `OA_HOME` variable to override the default behavior causes SiP Layout and Cadence I/O Planner to use that same OA runtime environment instead of the OA runtime environment that was built in their respective installations. Therefore, setting `OA_HOME` is not recommended, unless there is a compelling reason to do so.

NOTE: *If you use custom batch files or scripts to set up the SPB environment, add the following to the path. Some Allegro products may not launch without this setting.*

```
%CDSROOT%\OpenAccess\bin\win32\opt
```

SPB 16.3 README for SiP and IC Packaging

SPB 16.3 SiP/Cadence IO Planner Interoperability

The following table describes the versions of SiP and Cadence IO Planner validated by Cadence.

SiP/Cadence IO Planner Interoperability

SiP Version	Cadence IO Planner Version	OA Version
SPB16.30.000 Base	EDI09.10.000 ^{NOTE1}	OA 22.04.052
SPB16.20.019 Hotfix	SOC08.10.002	OA 22.04.044
SPB16.20.011 Hotfix	SOC08.10.001	OA 22.04.037
SPB16.20.006 Hotfix	SOC08.10.000	OA 22.04.032
SPB16.20.008 Hotfix	SOC07.10.003	OA 22.04.026
SPB16.20.000 Base	SOC07.10.002	OA 22.04.017
SPB16.01.041 Hotfix	SOC08.10.002	OA 22.04.044
SPB16.01.033 Hotfix	SOC08.10.001	OA 22.04.037
SPB16.01.024 Hotfix	SOC08.10.000	OA 22.04.032
SPB16.01.031 Hotfix	SOC07.10.003	OA 22.04.026
SPB16.01.011 Hotfix	SOC07.10.002	OA 22.04.017
SPB16.01.006 Hotfix	SOC07.10.001	OA 22.04.010
SPB16.01.000 Base	SOC07.10.000	OA 22.04.006
NOTE1: SPB 16.30.000 Base is validated with a pre-released version of EDI09.10.000. Validation with the EDI09.10.000 Base release will be completed in Q1 2010.		

SPB 16.3 README for SiP and IC Packaging

Cadence SiP RF Installation and Setup

Virtuoso Analog Design Environment (ADE) and Virtuoso Schematic Editor must be installed as part of the SiP RF Architect XL integrated solution. The licenses required for IC 6.1.3 and IC 5.1.41 with SPB 16.3 are as follows:

SiP RF Architecture on IC 6.1.3 (OA) Release and SPB 16.3		
SPB Licenses	SiP410	Cadence SiP RF Architect XL
	SiP225	Cadence SiP Layout XL
	SiP625	Cadence Chip Integration Option
Virtuoso Licenses	95100	Virtuoso® Schematic Editor L
	95200	Virtuoso® Analog Design Environment L
	38500	Virtuoso® Spectre® Simulator (with MMSIM 7.0.x or later release)
	95300	Virtuoso_Layout_Suite_L

SiP RF Architecture on IC 5.1.41 (CDBA) Release and SPB 16.3		
SPB Licenses	SiP410	Cadence SiP RF Architect XL
	SiP225	Cadence SiP Layout XL
	SiP625	Cadence Chip Integration Option
Virtuoso Licenses	34500	Virtuoso® Schematic Editor
	34510	Virtuoso® Analog Design Environment
	32500	Virtuoso® Spectre® Simulator
	300	Virtuoso® Layout Editor

When using Virtuoso IC 6.1.3 (OA database), set the following paths:

```
set path = (  
$ICHIER/tools/bin  
$ICHIER/tools/dfII/bin  
$PCBHIER/tools/dfII/bin  
$PCBHIER/tools/pcb/bin  
$MMSIMHIER/tools/bin  
$PCBHIER/tools/bin  
$PCBHIER/tools/fet/bin  
)
```

where

PCBHIER is SPB16.3 installation

ICHIER is IC613 installation

MMSIMHIER is MMSIM7.11.x installation (optional)

SPB 16.3 README for SiP and IC Packaging

Set the following paths if using Virtuoso IC 5.1.41 (CDBA database):

```
set path = (  
$ICHIER/tools/bin  
$ICHIER/tools/dfII/bin  
$PCBHIER/tools/dfII/bin  
$PCBHIER/tools/pcb/bin  
$PCBHIER/tools/bin  
$PCBHIER/tools/fet/bin  
)
```

where

PCBHIER is SPB16.3 installation

ICHIER is IC5141 installation

NOTE: To run SiP RF Architect, run *rwb*

Symbol Library and Padstacks

All library symbols created with previous 16.x releases of Allegro Package Designer (APD) are compatible with this release. Before loading the generic library, ensure that the customized component symbols are not in the same directory as the one where you load APD. This is to eliminate the possibility of customized library being overwritten when you load the generic APD library. The CAE libraries will not be affected. Before using any of the APD library symbols, carefully review them to ensure that they meet your physical design criteria.