cadence®

Virtuoso Abstract Generator User Guide

Product Version IC23.1
November 2023

© 2023 Cadence Design Systems, Inc.
Printed in the United States of America.

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

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.

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. 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.

Cadence is committed to using respectful language in our code and communications. We are also active in the removal and replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

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

1
Virtuoso Abstract Generator 13
Prerequisites for Virtuoso Abstract Generator 14
Abstract Generator Licensing Requirements 15
Abstract Generation Use Model 17
Support for Overridden and Instance Terminals 18
Flows Supported by Abstract Generator 20
GDSII to LEF 20
DEF to OA 20
LEF to OA 21
Virtuoso Layout Flow 22
Regular Expressions in Abstract Generator 24
Support for Special Characters in Regular Expressions 25
Support for Special Constructs “\” in Regular Expressions 28

2
Standalone Abstract Generator 31
Standalone Abstract Generator Environment Setup 32
Launching the Standalone Abstract Generator 33
Standalone Abstract Generator Interface 36
Customization of the Abstract Generator Interface 41
Custom Fonts in Standalone Abstract Generator Interface 41
Customized Appearance of the Standalone Abstract Generator Interface 44
Customization of the Abstract Generation Flow 51
Abstract Generation Flow 55
Import Data in Abstract Generator 56
Generate Abstracts 57
Inspect the Results of Generated Abstracts 58
Export Abstracts 59
Generating Abstracts 61
File Menu in Standalone Abstract Generator 63

© 2023 Cadence Design Systems, Inc.

Creating a Library in Abstract Generator . . . 64
Opening a Library in Abstract Generator . . . 66
Troubleshoot Library Issues . . . 67
Attaching Technology Information to a Library in Abstract Generator . . . 68
Troubleshoot Technology Issues . . . 69
Importing GDSII Information in Abstract Generator . . . 72
Importing OASIS Data in Abstract Generator . . . 74
Importing LEF in Abstract Generator . . . 76
Importing DEF in Abstract Generator . . . 79
Importing Logical Data in Abstract Generator . . . 80
Importing Options File in Abstract Generator . . . 81
Creating an Abstract from a DEF Block . . . 82
Exporting LEF in Abstract Generator . . . 84
Troubleshoot Import and Export Issues . . . 86
Specifying Global Options in General Options form . . . 91
Viewing the Statistics Report in Abstract Generator . . . 93
Bin Preferences for Abstract Generation . . . 95
Creating a User Bin in Abstract Generator . . . 97
Adding a User Bin in Abstract Generator . . . 98
Renaming a User Bin in Abstract Generator . . . 99
Deleting a User Bin in Abstract Generator . . . 100
Setting Bin Options in Abstract Generator . . . 101
Guidelines for Bin Options Form . . . 104
Troubleshoot System Bin Issues . . . 107
Cell Preferences for Abstract Generation . . . 109
Selecting Cells Based on a Criteria in Abstract Generator . . . 112
Revalidation of Cells with Invalid Views in Abstract Generator . . . 114
Sorting Cells in Abstract Generator . . . 115
Distributing Cells into Bins in Abstract Generator . . . 116
Moving Cells Across Bins in Abstract Generator . . . 119
Viewing and Editing Cell Properties . . . 120
Viewing and Editing Cell Terminal Properties in Abstract Generator . . . 123
Support for Inherited Connections in Abstract Generator . . . 125
Viewing a Cell Status Report in Abstract Generator . . . 128
Editing Cellviews in Virtuoso Layout Editor in Abstract Generator . . . 129
Troubleshoot Cell Issues . . . 131

© 2023 Cadence Design Systems, Inc.

Abstract Generation in Batch Mode . . . 133
Abstract Generator Log, Replay, and Record Files . . . 135
Specifying the CDS.log File Name . . . 137
Message Filters in Abstract Generator . . . 138
Exporting a Report in Abstract Generator . . . 139
Recording Commands and Events in Abstract Generator . . . 140
Replaying Commands in Abstract Generator . . . 142
Conversion of Common Lisp Files to SKILL or TCL Format . . . 143
Options and Status Files in Abstract Generator . . . 145
Exporting Options File in Abstract Generator . . . 147
Guidelines for Optimizing Abstract Generator Performance . . . 148

3
Pins Step in Standalone Abstract Generation . . . 151
Specifying Pin Mappings for Abstract Generation . . . 153
Text Labels-to-Pins Mapping . . . 155
Regular Expressions for Creating Pin Names . . . 159
Pin Names . . . 160
Customizing Text Labels for Abstract Generation . . . 162
Setting PR Boundary Creation Options in Standalone Abstract Generation . . . 165
Customizing Abstracts of Routed Blocks in Standalone Abstract Generator . . . 168
N-Well and P-Well Pins Creation . . . 170
N-Well Pins . . . 170
P-Well Pins . . . 171
Multiple N-Well and P-Well Pins . . . 172
Troubleshoot Pin Issues . . . 175

4
Extract Step in Standalone Abstract Generation . . . 183
Specifying Layers and Geometries for Extraction . . . 186
Layer Assignment Tables . . . 189
Specifying General Options in G-Spec Builder Form . . . 192
Relationship between Signal and Antenna Tabs . . . 196
Must-Join and Must-Connect Relationships for Terminals . . . 199
Antenna Generation for MUSTJOIN and MUSTJOINALLPORTS Terminals . . . 200

© 2023 Cadence Design Systems, Inc.

Extracting Power Nets in Standalone Abstract Generator . . . 209
Processing Antenna Information in Standalone Abstract Generator . . . 212
FinFET Support in Abstract Generator . . . 215
FDSOI Support in Abstract Generator . . . 216
Antenna Calculation by Abstract Generator . . . 221
Calculation of ANTENNAMAXCUTCAR . . . 224
Keywords Used in Antenna Calculation . . . 227
Checks Performed During Antenna Calculation . . . 228
Comparison of Geometry Specifications for Antenna Regions and Antenna Extraction 231
Specifying Connectivity Information in Designs . . . 232
Restrictions for Signal and Power Pin Geometry . . . 235
Troubleshoot Extract Issues . . . 236

5
Abstract Step in Standalone Abstract Generation . . . 241

Customizing Pin Shapes in Standalone Abstract Generator . . . 243
Square and Non-Square Boundary Pins . . . 247
Specifying Power and Ground Rail Characteristics . . . 250
Power Rail Scenarios . . . 254
Abstracts for Routed Blocks with Power Rings . . . 260
Boundary Pin Trimming . . . 262
Modifying Blockage Geometry in the Abstract View . . . 264
Performance Improvements for Generating Detailed Blockages . . . 266
Control Spacing Value During Geometry Add Backs . . . 268
Impact of Cutout and Spacing Options on Blockage Types . . . 269
Pin Cutout Support on Metal Layers Above the Pin Layer . . . 273
Routing Channel Creation when Generating Cover Blockages . . . 274
Automatic Pin Stretching . . . 276
Calculating Metal Layer Density . . . 279
Density Calculation . . . 281
Fracturing Pins and Blockages . . . 285
Pin and Blockage Fracturing . . . 287
Specifying Sites in Standalone Abstract Generator . . . 290
Creating an Overlap Boundary . . . 293

© 2023 Cadence Design Systems, Inc.

Retain Layout Information in the Abstract View . . . 295
Troubleshoot Abstract Messages . . . 297
Creation of Fill Blockages on Specific Layers . . . 306
Bus Annotation in Abstract Generator . . . 308
Support for EXCEPTPGNET LEF Attribute . . . 310
Layout Dual View Support . . . 311
Abstract Data Annotated in Layout Dual View . . . 312
Troubleshoot Layout Messages . . . 314
Support for Copying Mixed Signal Routing Constraints from Layout to Abstract View . 316

6
Integrated Abstract Generator . . . 319
Specifying Design Information in the Abstract Generator . . . 322
Generating Abstracts . . . 325
Viewing Results in the Abstract Generator Form . . . 329
Viewing the Cell Report . . . 329
Viewing Log Information . . . 330
Viewing the List of Cells . . . 331
Customizing Abstract Generator Flow Using SKILL Files . . . 333

7
Dynamic Abstract Generation . . . 335
Prerequisites for Dynamic Abstract Generation . . . 337
Dynamic Abstract Generation Use Model . . . 339
Dynamic Abstract Generation Commands . . . 341
Abstracting Pcells . . . 342
Loading and Saving an Options File for Dynamic Abstract Generation . . . 346
Unabstracting Pcells . . . 348
Highlighting Unabstracted Pcells . . . 349
Marking or Unmarking Pcells for Abstraction . . . 350
Specifying Advanced Dynamic Abstraction Rules . . . 351
Specifying General Dynamic Abstraction Options . . . 353
Specifying Extraction Options for Dynamic Abstraction . . . 356
Specifying Antenna Options for Dynamic Abstraction . . . 360

© 2023 Cadence Design Systems, Inc.

8
Multi-Patterning Technology Support in Abstract Generator 363

Prerequisites for Multi-Patterning Technology Support 365
Color Representation for Pins 367
Pins Step 367
Extract Step 368
Abstract Step 370
Color Representation for Blockages 372
Cover Blockage 372
Shrink Blockage 372
Detailed Blockage 373
Supported Blockage Colors in Multi-Patterning Technology 374
Support for Trim Metal 376

A
Virtuoso Abstract Generator Forms 379

Abstract Generator Form 381
Abstract Pcells Form 384
Add User Bin Form 387
Attach Technology Library Form 388
Cell Properties Form 389
Cell Report Window 390
Copy Bin Options Form 391
Delete User Bin Form 392
Distribute Cells Among Bins Form 393
Export LEF Form 395
Export Report Form 397
Export Options Form 398
General Options Form 399
General 399
Views 402
G-Spec Builder Form 403
Import DEF Form 405
Import Layout Form 406

© 2023 Cadence Design Systems, Inc.

Import LEF Form . . . 408
Import Logical Form . . . 410
Import Oasis Form . . . 411
Import Options Form . . . 413
Integrated Abstract Generator Form . . . 414
Mark/Unmark Pcells for Abstraction Form . . . 418
Move Selected Cells Form . . . 419
New Library Window . . . 420
Open Library Window . . . 421
Rename User Bin Form . . . 422
Replay Form . . . 423
Running Step Pins Form . . . 424
- Map Tab . . . 425
- Text Tab . . . 427
- Boundary Tab . . . 429
- Blocks Tab . . . 431

Running step Abstract Form . . . 433
- Adjust . . . 434
- Blockage . . . 447
- Density . . . 453
- Fracture . . . 455
- Site . . . 457
- Overlap . . . 458

Running Step Extract Form . . . 462
- Signal Tab . . . 463
- Power Tab . . . 470
- Antenna Tab . . . 472
- General Tab . . . 483

Save Dynamic Abstract Generation Options Form . . . 485
- General . . . 486
- Extraction . . . 487
- Antenna . . . 489

Select Cells Form . . . 491
Sort Cells Form . . . 492
Specify SKILL Files Form . . . 493
UnAbstract Pcells Form . . . 494

© 2023 Cadence Design Systems, Inc.

B
Virtuoso Abstract Generator Environment Variables 495
List of Abstract Generator Environment Variables 495
optionsFile 496
showAbstractSection 497
showAntennaOptions 498
showExtractGspecTables 499
showExtractSection 500
showPinsSection 501
showOverlapBoundaryOption 502
showPinGroupingOptions 503

C
Abstract Generator Menu Commands 505
Standalone Abstract Generator Menu Bar 506
Dynamic Abstract Generator Context-Sensitive Menu 508
Layout Canvas Using the Context-Sensitive Menu 510

D
Library Design Guidelines 511
Guidelines for Designing a Cell Library 513
Guidelines for Designing Pins 517
Design IO, Corner Cells, and Blocks 523
Guidelines for Setting Pitch in a Cell Library 524
Guidelines for Setting Pitch for Layers with the Same Preferred Direction 528
Guidelines for Setting Pitch for Via Stacking 530
Guidelines for Designing a Gridded Library for Innovus 531
Guidelines for Converting an Existing Library into Gridded Library 532
Guidelines for Modeling Pins in a Cell Library 533
Power to Blockage Violations in Innovus 534
Virtual Tracks in Innovus 535
Guidelines for Centering On-grid Pins in Innovus 536
Routing Issues in Pins 537
Guidelines for Defining Blockages in a Cell Library 538

© 2023 Cadence Design Systems, Inc.

Pin Accessibility of Blockages in Abstract Generator . . . 540
On-grid Versus Off-Grid Blockages in Abstract Generator . . . 541
Guidelines for Setting Cell Porosity and High Density in a Cell Library . . . 542
Pitch Routing in Abstract Generator . . . 543
Pin Spacing in Abstract Generator . . . 545
Pin Staggering in Abstract Generator . . . 546
Tracks for Wide Wire Designs . . . 547
Variable-Height and Multiple-Height Cells . . . 548
Non-rectangular Blocks . . . 552

E
Technology File Requirements for Abstract Generation . . . 553
Modification of Technology Information . . . 554
Technology File Requirements for Virtuoso Abstract Generator . . . 556
Devices . . . 556
Layer Definitions . . . 557
Layer Names . . . 560
Via Definitions . . . 566
Physical Rules . . . 567
Voltage Dependent and Layer-Purpose-Based Rules . . . 571
Defining a Default Via in the OpenAccess Technology File . . . 573
Requirements for Antenna Calculation . . . 574
Requirements for Metal Density Calculation . . . 578
Requirements for Viewing Data In Layout . . . 579
Requirements for Specifying Place-and-Route Information . . . 580
Troubleshoot Technology File-Related Issues in the Virtuoso Studio Design Environment 583
Troubleshoot Metal Layer Issues . . . 585
Troubleshoot Layer Spacing Rules Issues . . . 587
Troubleshoot Via Layers Issues . . . 588
Troubleshoot Routing Layer Issues . . . 590
Troubleshoot Minimum Width Value Issues . . . 592
Troubleshoot Minimum Spacing Value Issues . . . 594
Troubleshoot Cut Layer Issues . . . 595
DATABASE MICRONS in the Exported LEF File . . . 596

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

1

Virtuoso Abstract Generator

Abstract Generator is a library modeling tool that lets you create abstracts for standard cells, macro blocks, and IOs from detailed layout information in Library Exchange Format (LEF), Design Exchange Format (DEF), and Graphics Design Station II (GDSII) Stream formats.

Important

LEF/DEF version 5.8: References to LEF and DEF version 5.8 functionality and features apply only for Abstract Generator in the Innovus™ Digital Implementation System (Innovus) releases. LEF/DEF 5.8 is not available with Innovus.

An abstract is a high-level representation of a layout view, which contains information about the type and size of cells, position of pins or terminals, and the overall size of blockages. The generated abstracts are based on physical layout and logical data, process technology information, and specific cell-modeling requirements. The abstracts are used in place of full layouts to improve the performance of place-and-route tools, such as Innovus. After the place-and-route is complete, the abstracts are replaced back with the layouts.

The functionality for abstract generation is available through two interfaces, which access the same executable. You can choose to access a particular interface based on your requirements, the current design environment, and the familiarity with an interface. You can invoke the executable either as a standalone application or from within the Virtuoso Studio design environment.

- The standalone Abstract Generator interface can be used to generate abstracts for a single library at a time. You can run this application both in GUI and non-GUI mode. The graphical interface of this application is modeled on the abstract generation flow steps — *Pins*, *Extract,* and *Abstract*. This interface provides advanced options and is designed for experienced users.
- The integrated Abstract Generator interface launched from the Virtuoso Studio design environment can be used to generate abstracts for individual or multiple cells of a library. This interface is simple and is designed for novice users who are either not familiar with or find the advanced options of the standalone application difficult to comprehend. Being tightly integrated with the Virtuoso Studio design environment, this interface allows users

© 2023 Cadence Design Systems, Inc.

working on layouts to continue with generating abstract for their layouts in the same environment.

- The Dynamic Abstract Generator is used by Virtuoso Space-Based Router. It automatically creates abstracts for the Pcells based on a set of abstraction rules.

Prerequisites for Virtuoso Abstract Generator

This section is aimed at developers and designers of integrated circuits and assumes that you are familiar with:

- The Virtuoso Studio design environment and application infrastructure mechanisms designed to support consistent operations between all Cadence® tools.
- The applications used to design and develop integrated circuits in the Virtuoso Studio design environment, notably, the Virtuoso Layout Suite, and Virtuoso Schematic Editor.
- The Virtuoso Studio design environment technology file.
- Component description format (CDF), which lets you create and describe your own components for use with Layout XL.

Related Topics

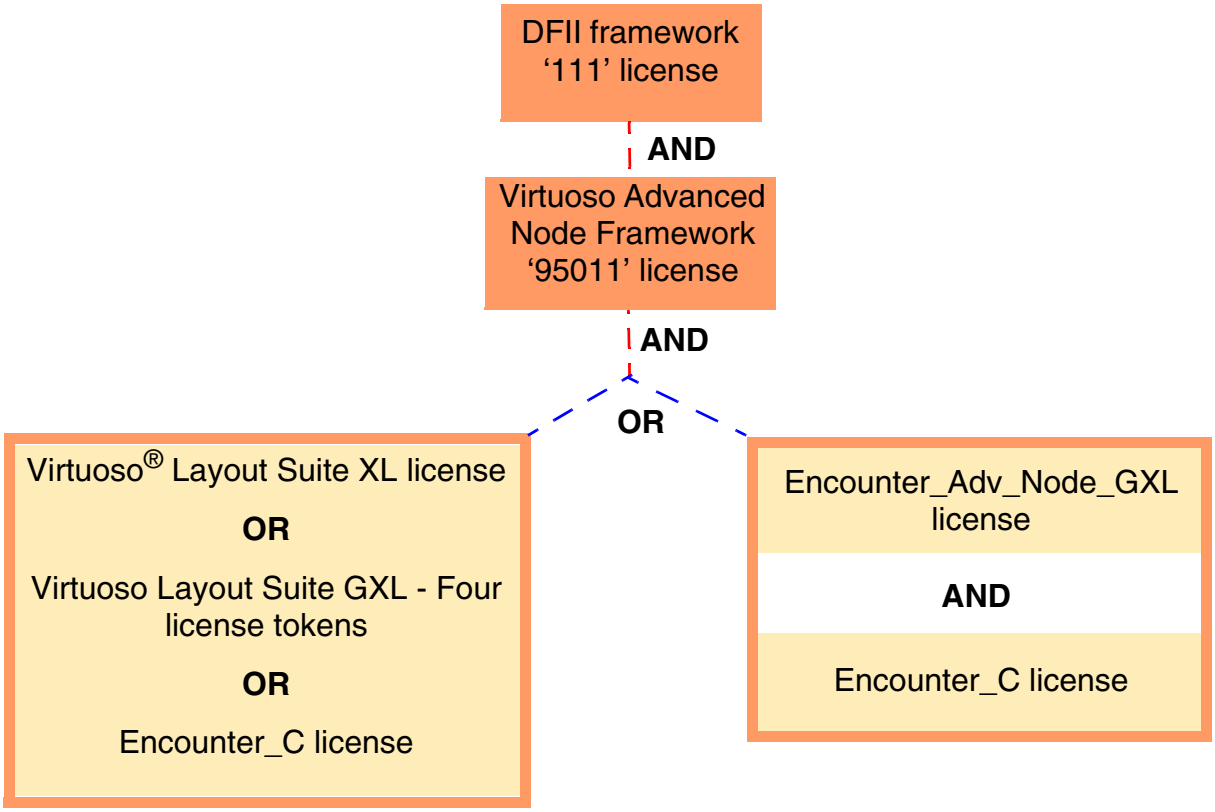
Configuring the Virtuoso Studio Design Environment

© 2023 Cadence Design Systems, Inc.

Abstract Generator Licensing Requirements

Standalone Abstract Generator

The following figure describes the licensing requirements for standalone Abstract Generator:



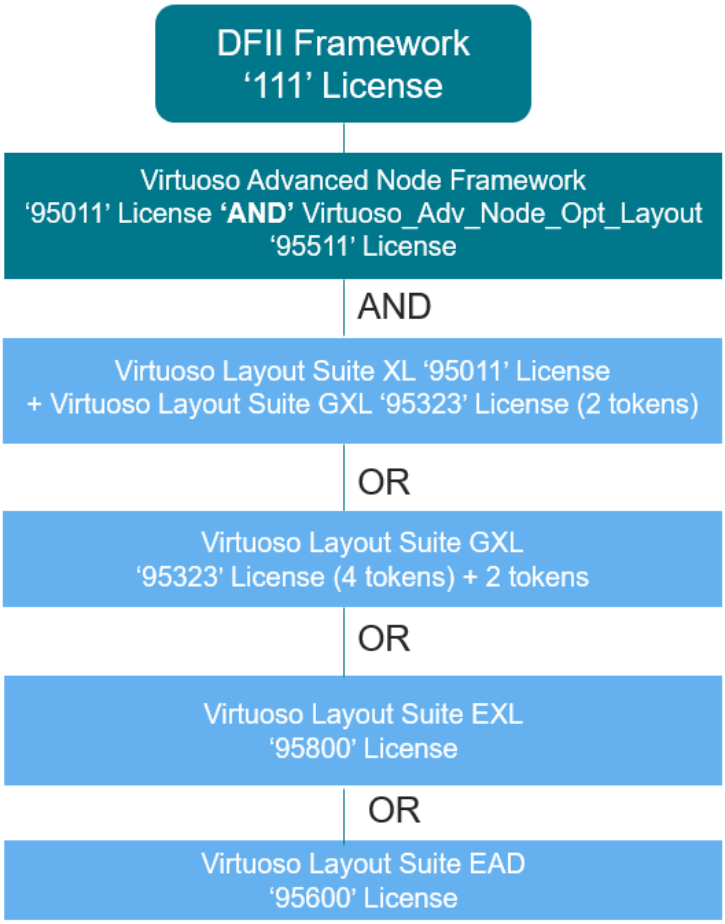
Integrated Abstract Generator

Required licenses:

- DFII framework '111' license
- Virtuoso Advanced Node Framework '95011' license
- One of the following:
  - Virtuoso Layout Suite XL license + 2 GXL tokens
  - Virtuoso Layout Suite GXL (4 tokens) + 2 GXL tokens
  - Virtuoso Layout Suite EXL

© 2023 Cadence Design Systems, Inc.

The following image captures the licensing behavior of the integrated Abstract Generator.



If Abstract Generator has checked out one of the above mentioned license, then it does not check out any other additional licenses.

Additionally, when a library is opened, if the technology file has data reflecting geometries of 20nm, 16nm, or 14nm, the `Virtuoso_Adv_Node_Opt_Lay_Std` license (95512) is checked out. If the technology file has data reflecting geometries of 10nm or smaller, the `Virtuoso_Adv_Node_Opt_Layout` license (95511) is checked out.

Related Topics

Abstract Generation Flow

Abstract Generation Use Model

© 2023 Cadence Design Systems, Inc.

Abstract Generation Use Model

Abstract Generator generates LEF abstracts for cells in cell libraries. A cell library typically consists of standard cells but can also include IO and corner cells and some small custom blocks. The individual components of the library are described in an appropriate layout format, such as GDSII or LEF.

Abstract Generator lets you import layout information and create abstracts for some or all of the cells in the library. You can then reference any of the library cells in your design, safe in the knowledge that a suitable abstract exists for that cell. After generating abstracts for library cells, you should not have to return to the tool unless the library is revised either in response to some change in the fabrication process or through the addition of new cells or modification of the existing ones.

Abstract Generator satisfies the requirements of library designers and production engineers by presenting an uncluttered flow through the library development process.

The more experienced user enjoys maximum flexibility in entering cell information, partitioning cells into exclusive sets, fine tuning option settings, and iterating through the abstract generation process. The tool also provides reliable support for moving backward in the flow and clearing feedback on potential and real problems with the abstracts generated.

The less experienced designer is provided with a simple flow through the essential steps required to update a library with predefined settings. The more flexible and user-friendly interface, that is, the integrated Abstract Generator, which can be launched in the Virtuoso Studio design environment is also available at the disposal of the novice users.

Related Topics

Flows Supported by Abstract Generator

© 2023 Cadence Design Systems, Inc.

Support for Overridden and Instance Terminals

Abstract Generator supports overridden terminals and instance terminals in block domain. The Abstract Generation commands do not create overriding terminals or instance terminals. However, these overridden terminals and instance terminals can be present in OpenAccess data coming from Innovus or any other application using OA. Therefore, the support is required for interoperability between Innovus and Virtuoso.

The behavior of commands in Abstract Generator is given as follows:

- The commands processing pins and pin figures on overridden terminals and instance terminals, of master instance having pins and pin figures, should consider overridden terminal and instance terminal as a normal block domain terminal or instance terminal, without any difference in the behavior. For example, the Pin Optimize command should optimize pins on overridden terminal and instance terminal.
- The commands involving change of signal type should ensure that nets connected to overriding terminals and instance terminals can only be of signal type `power`, `ground`, `tieHi`, `tieLo`, and `tieOff`.

  In Abstract Generator, you can set the signal type for terminals only as `signal`, `clock`, `power`, `ground`, or `analog`. If pins on the layout and the terminals for the same pins have another signal type, such as `reset`, the signal type value is retained as `reset` in the abstract view as well.
- The commands processing (deleting) empty nets should be careful because a net may be empty only in the block domain and may have a terminal/instance terminal in module domain. Therefore, the net is not empty as corresponding terminals and instance terminals are hidden in the block domain.

Command related to terminals and instance terminals is *Abstract Generate*.

The `tieHigh`, `tieLow`, `tieOff`, `power`, `ground` and `signal` nets specified in verilog can be implemented in block domain using any physical-only net. The example for this is given as follows:

```
/ Verilog file for cell "top_level" view "layout"
// Language Version: 2001
module top_level (
    A,
    B,
    C,
    D,
    Z
    );
```


© 2023 Cadence Design Systems, Inc.

```
    input A;
    input B;
    input C;
    input D;
    input _IN;
    output Z;
    output _OUT;
    wire net1;
    assign _OUT = 1'b1;
    assign _IN = 1'b0;
    block3 I1 ( .A(A), .B(B), .C(C), .Y(net1), .Z(1'b1));
    block I2 ( .A(net1), .B(1'b1), .C(D), .Z(Z), .„nd! (1'b0), .\vdd! (1'b1));
endmodule // top_level
```

In the mentioned example, observe the following in the connectivity:

- The terminals `_IN` and `_OUT` are connected to `tie0` and `tie1` nets, respectively.
- The instance terminals corresponding to `I1/Z` and `I2/B` are connected to tie.
- The instance terminals corresponding to `I2/vdd!` and `I2/gnd!` are connected to `tie1` and `tie0` nets, respectively

It is significant to understand that verilog designer has no clue how 1'b1 (`tieHi`) and `1'b0` (`tieLow`) nets will be implemented in the layout or block domain. It depends on the block designer to choose any physical only net to implement `1'b1` and `1'b0`. For example, the following array of nets can be used to implement.

- `_IN` and `I2/gnd!` (`tie0`) to be implemented by net `VSS` (physical only).
- `_OUT` and `I2/vdd!` (`tie1`) to be implemented by net `VDD` (physical only).
- `I1/Z` and `I2/B` (`tie1`) to be implemented by net `INTERNALVDD` (physical only).

The following changes in the block domain will have to be made.

- Hide `_IN _OUT` terminals.
- Hide instance terminals `I2/gnd!`, `I2/vdd!`, `I1/Z` and `I2/B`.
- Create overriding terminals and overriding instance terminals with same names to implement them using the above mentioned physical only nets.

Related Topics

Flows Supported by Abstract Generator

© 2023 Cadence Design Systems, Inc.

Flows Supported by Abstract Generator

The following flows describe the interaction between the standalone Abstract Generator interface, and the tools and translators on the Virtuoso Studio platform:

- GDSII to LEF
- DEF to OA
- LEF to OA
- Virtuoso Layout Flow

GDSII to LEF

In this flow, Abstract Generator imports GDSII data and generates the abstract LEF 5.x. This LEF abstract is then used to create layouts, which can be imported directly into the Virtuoso environment.

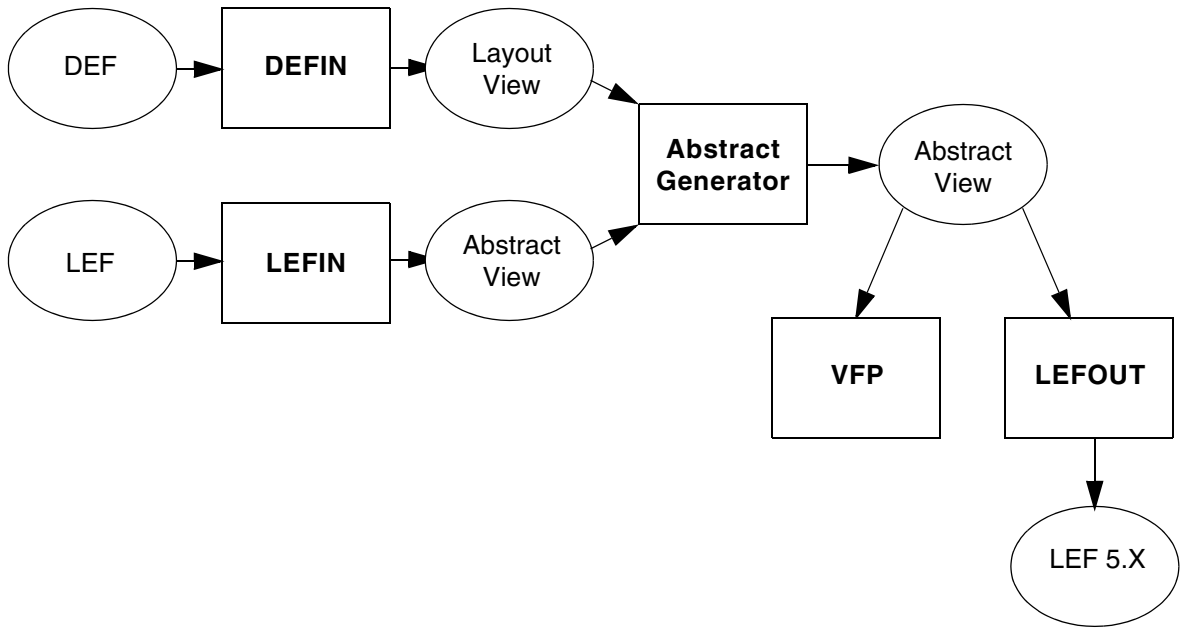
The abstract generation step is usually performed by a library or an IP developer and is independent of any floorplanning.

DEF to OA

In this flow, Abstract Generator imports DEF layout and LEF abstract views and generates an abstract layout. The imported LEF is required to create the abstract views of cells, which are then instantiated when the DEF is read by Abstract Generator.

© 2023 Cadence Design Systems, Inc.

This flow lets you convert a routed block generated by Innovus into a cellview suitable for use in the Virtuoso Studio design environment. You can also generate a LEF version of the block abstract if required.



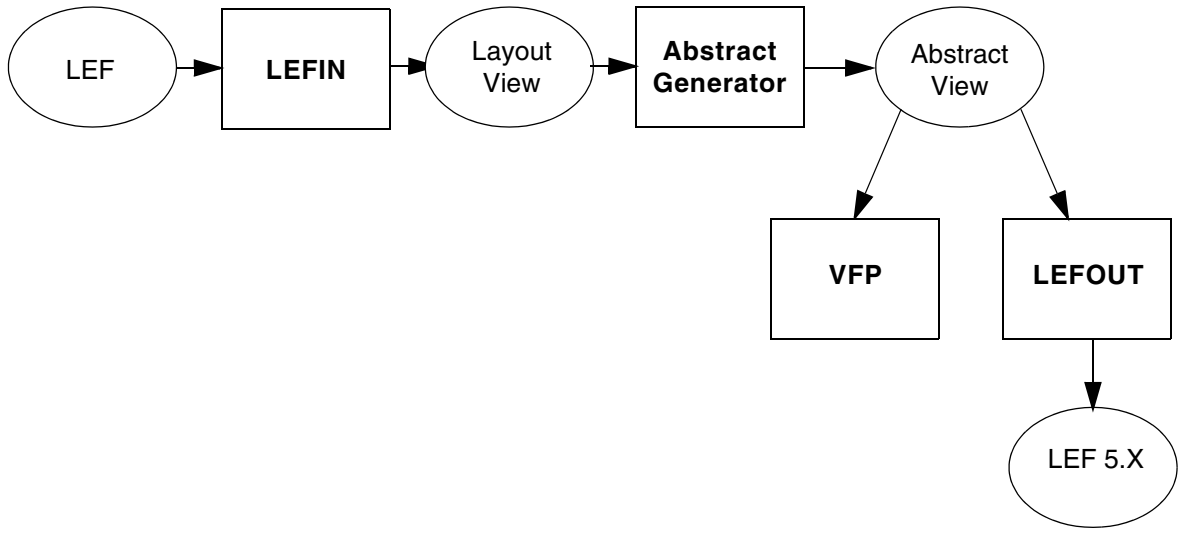
LEF to OA

In this flow, LEF layout information is imported to generate an abstract layout view.

This flow lets you import an existing LEF abstract generated by a non-Cadence tool and use the advanced features of Abstract Generator to render the data for use in Cadence floorplanning and place-and-route engines, particularly power routers. For example, you might use Abstract Generator to form correct power geometry groupings into `LEF PORTS` for `sroute`.

© 2023 Cadence Design Systems, Inc.

The abstract view can be used directly in the Virtuoso Studio design environment or you can export LEF 5.x for use with Innovus.



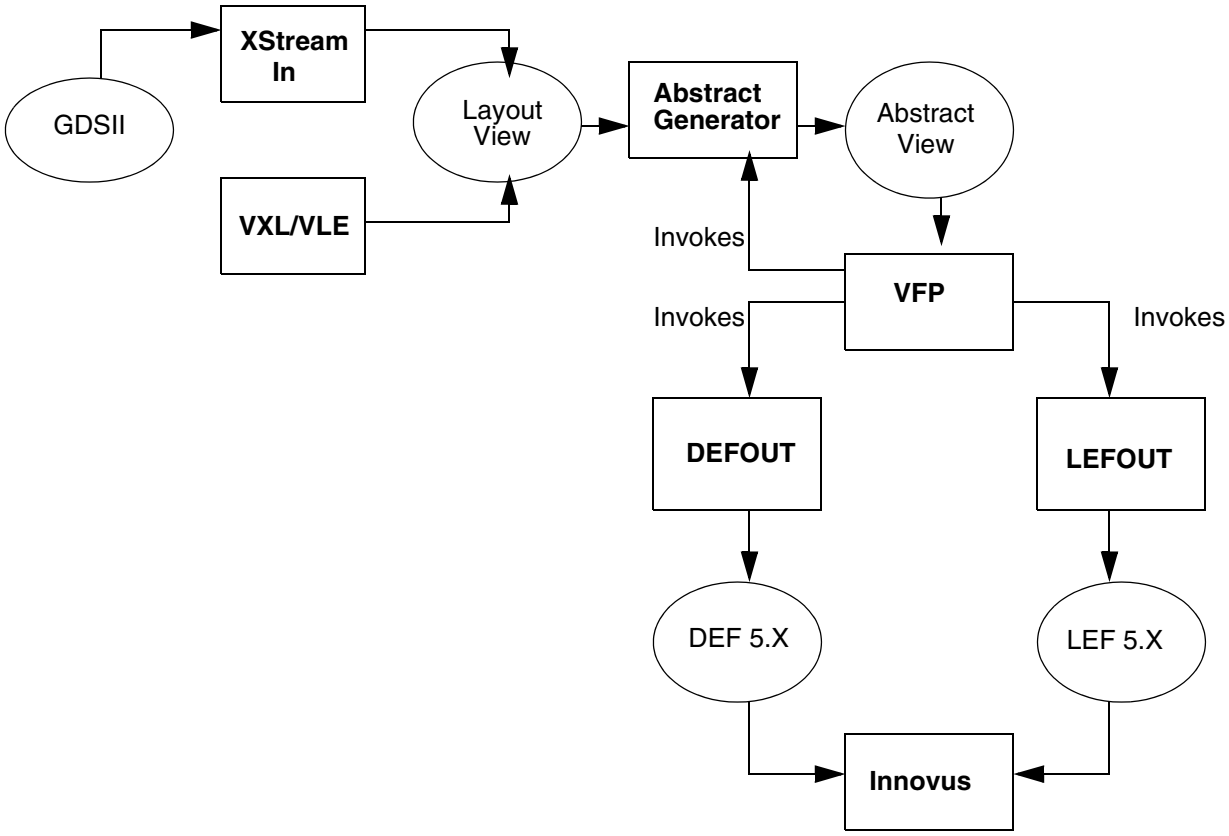
Virtuoso Layout Flow

This flow facilitates a tight integration between Abstract Generator and Virtuoso Floorplanner, which bypasses the import and export LEF steps used in the LEF to OA flow.

In this flow, all layout data creation and abstract generation stay within the Virtuoso Studio design environment. There is no requirement for LEF. This part of the flow can be done by using Virtuoso Floorplanner or Abstract Generator. The DEF side of the flow requires a netlist and can only be done by using Virtuoso Floorplanner.

© 2023 Cadence Design Systems, Inc.

Virtuoso Floorplanner can invoke Abstract Generator to generate an abstract view for either a layout or a softLayout view.



Related Topics

Customizing Pin Shapes in Standalone Abstract Generator

© 2023 Cadence Design Systems, Inc.

Regular Expressions in Abstract Generator

This topic contains guidelines on how to use regular expressions in Abstract Generator. This information is useful if you use regular expressions to complete particular entry fields in the Abstract Generator GUI. Abstract Generator does not check the entered regular expressions against any syntax rules.

Regular expression are strings of characters used for pattern matching. The characters in a regular expression are classified as either special or ordinary.

This topic lists the special characters and describes how to use them. Any character in a regular expression that is not special is considered ordinary, unless preceded by the character \. You can concatenate any two regular expressions to produce a new regular expression. For example, concatenating the regular expressions `f` and `i` produces a regular expression that matches the string `fi`. By incorporating special characters, you can create more sophisticated regular expressions that matches variety of strings.

Related Topics

Support for Special Constructs “\” in Regular Expressions

Support for Special Characters in Regular Expressions

© 2023 Cadence Design Systems, Inc.

Support for Special Characters in Regular Expressions

A description of each special character and its intended use is detailed below:

Special Character	Description of Use
. (period)	This is used as a special character to match any single character except a new line. If you use concatenation, you can construct regular expressions such as `x.y`, which will match any three character string beginning with an `x` and ending in a `y`.
*	This special character is not a construct in itself; it is a suffix, which means that the preceding regular expression should be repeated as many times as possible. For example, in `go*`, the `*` applies to the `o`; so `go*` would match one `g` followed by any number of `o`s. Thus, `go` has a repeating `o`, but not a repeating `go`. The string matcher processes a `*` construct by matching as many repetitions that can be found. It will then continue with the rest of the pattern entered. If this fails to find a match, it backtracks, discards some of the matches of the `*` construct, and again tries to match the rest of the pattern. For example, matching `ca*ar` against the string `caaar`, the `a*` first tries to match all three `a`s, but the rest of the pattern is `ar` and there is only `r` left to match, so this attempt fails. The next alternative match is for `a*` to match only two `a`s. With this last choice, the regular expression matches successfully.
+	This is another suffix character, similar to `*`, except that it requires that the preceding expression be matched at least once. For example, `ca+r` will match the string `car` and `caaar`, but not the string `cr`, whereas `ca*r` would match all three strings.
?	This is another suffix character similar to `*`, except that it can match a preceding expression once or not at all. For example, `ca?r` will match `car` or `cr`, but nothing else.

© 2023 Cadence Design Systems, Inc.

`[ ... ]`

The `[` begins a *character set*, which is terminated by the `]`. Using its simplest case, the characters between the two form the character set. Therefore, `[ad]` matches either one `a` or one `d` and `[ad]*` matches any string composed of just `a`s and `d`s.

You can also specify *character ranges* within a character set. This is done by inserting a hyphen (`-`) between the first and last characters in the range. For example, `[a-z]` will match any lower case letter. You can also use ranges in conjunction with any individual characters, for example `[a-z$*]`, which will match any lower case letter or a `$` or a `*`.

Note: Any special characters contained inside a character set are treated as ordinary. A new set of special character exists inside character sets, specifically `]`, `-`, and `^`.

If you want to include `]` as a special character in the character set, ensure it is the first character listed in the character set. For example, `[]a]` matches `]` or `a`.

If you want to include `-`, you must specify this as `---`, which is a range containing only `-`.

If you want to include `^`, you must make it any other character than the first character in the set. See `[^ ... ]` below for further information.

Using `^` at the beginning of a character set begins a *complement character set*. This allows you to match any characters except the ones specified. For example, [`^a-z0-9A-Z]` would match all characters except letters and digits.

The `^` is only a special character if it is the first digit in a character set.

`^`

When not part of a character set `^` is a special character that matches an empty string, but only if at the beginning of a line in the text being matched.

For example, `^foo` matches a `foo` that occurs at the beginning of a line.

© 2023 Cadence Design Systems, Inc.

`\`	The `\` special character has two functions; it quotes special characters (including `\`), and secondly it introduces additional special constructs. Because the `\` quotes special characters, `\$` is a regular expression that matches only `$`, and `\[` is a regular expression that matches only `[`, and so on. See Support for Special Constructs "\" in Regular Expressions for more information about the use of this character.
`$`	The placement of the `$` special character at the end of an entire regular expression constrains that regular expression to match a final segment of a line.

Examples

Regular expressions can be used when completing the names of the pins in the *Pins* step.

Examples of regular expression entries in the *Power pin names* field can include:

- `^ovdd$` which matches the text `ovdd`
- `^ovdd` which matches any text that begins with `ovdd`
- `ovdd` which matches any text containing `ovdd`
- `vdd*` which matches `vdd`, `vddd`, `vddd`,...
- `vdd.*` which matches `vdd` or `vddfred`

Related Topics

Pin Names

Regular Expressions for Creating Pin Names

Support for Special Constructs "\" in Regular Expressions

Specifying Pin Mappings for Abstract Generation

© 2023 Cadence Design Systems, Inc.

Support for Special Constructs "\" in Regular Expressions

In general, \ followed by any character matches only that character. There are, however, several exceptions to this rule, where characters preceded by \ are special constructs. Such characters are always ordinary when encountered on their own.

The following table describes possible \ special constructs.

\ Special Construct	Description of Use
|	This specifies an alternative. For example, `foo|bar` matches either `foo` or `bar` but no other string. Full backtracking capability allows you to handle multiple uses of |.
\(... \)	This grouping construct serves three purposes: **1.** To enclose a set of | alternatives for other operations. Therefore, `\(foo|bar\)x` matches either `foox` or `barx`. **2.** To enclose a complicated expression for the postfix * to operate on. Therefore, `ba\(na\)*` matches 'bananana' with any (zero or more) number of `na` strings **3.** To mark a matched substring for future reference. See *\n* below.
\n	After the end of a \(... \) construct, the matcher remembers the beginning and end of the text matched by that construct. Then, later on in the expression, you can use \ followed by *n* to mean "match the same text matched the *n*th time by the \(...\) construct."
\‘	This matches the empty string, provided it is at the beginning of the buffer.
\’	This matches the empty string, provided it is at the end of the buffer.
\B	This matches the empty string, provided it is not at the beginning or end of a word.
\<	This matches the empty string, provided it is at the beginning of a word.
\>	This matches the empty string, provided it is at the end of a word.

© 2023 Cadence Design Systems, Inc.

`\w`	This matches any word-constituent character determined by the editor syntax table.
`\W`	This matches any character that is not a word-constituent.
`\`*`scode`*	This matches any character whose syntax is code (a character that represents syntax code)
`\`*`Scode`*	This matches any character whose syntax is not code.

Related Topics

Support for Special Characters in Regular Expressions

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

2

Standalone Abstract Generator

The standalone Abstract Generator interface is designed to be sufficiently intuitive to allow first-time users to make good progress and at the same time provide the flexibility required to satisfy the goals of expert users. The interface provides comprehensive status information and supports the recommended use model by allowing backward steps in the flow and providing helpful feedback at all stages.

You can launch Abstract Generator and use the flow defined by the tool to generate abstracts from layout data. You can use the commands, options and graphical elements to describe the features that you encounter while using the interface. Some abstract generation functions can adversely affect the performance of the tool and should be run only when necessary. Use the guidelines to optimize the performance of Abstract Generator.

Related Topics

Launching the Standalone Abstract Generator

Standalone Abstract Generator Interface

Abstract Generation Flow

Customization of the Abstract Generator Interface

Guidelines for Optimizing Abstract Generator Performance

© 2023 Cadence Design Systems, Inc.

Standalone Abstract Generator Environment Setup

Abstract Generator reads a file named `.abstractrc` during the startup process. If you use the command line options to launch Abstract Generator, the tool first loads the `.abstractrc` file and then the replay file that has been specified using the `-replay` argument.

When the tool starts, it searches for the `.abstractrc` file at the following locations in the order specified:

- *`cds_install_dir`*`/tools/dfII/local`
- Current directory (.)
- Home directory (~)

You can modify the `.abstractrc` file to customize the tool for a group of users. You can also specify your custom standard cell SKILL code in the `.abstractrc` file so that Abstract Generator reads it every time you launch the tool. In addition, the `.abstractrc` file can enable you to run the required TCL or SKILL replay file when you launch the tool.

Do not set up library options in the `.abstractrc` file because they are overwritten when you open a library.

Related Topics

Launching the Standalone Abstract Generator

Standalone Abstract Generator Interface

© 2023 Cadence Design Systems, Inc.

Launching the Standalone Abstract Generator

You can launch the standalone Abstract Generator directly from the installation directory.

To launch the standalone interface:

1. Ensure that you have the following directory in your path:

   your_install_dir/tools/dfII/bin

2. Move into the run directory. You can run Abstract Generator from any directory, but preferably not from the release hierarchy.

3. Type `abstract` at the command line prompt. You can specify various arguments with the `abstract` command, such as:

   ```
   abstract
       [-replay filename]
       [-log filename]
       [-tcl]
       [-nogui]
       [-V | -W]
       [-dbType]
       [-cdslib path]
       [-help]
   ```

   These arguments are described below.

Argument	Description
`-replay` *`filename`*	Specifies a TCL or SKILL replay file to be run automatically when Abstract Generator launches. The format of the replay file depends on the extension language setting. By default, Abstract Generator operates in the SKILL mode.
`-log` *`filename`*	Specifies the name of the log file to be generated for the current session. By default, the log file is named `abstract.log`.
`-tcl`	Sets the extension language to TCL. This means that Abstract Generator recognizes only TCL commands in its command line and replay file. By default, Abstract Generator operates in the SKILL mode.
`-nogui`	Disables the graphical user interface. Abstract Generator operates directly from the command line prompt in the shell window.

© 2023 Cadence Design Systems, Inc.

`-V | -W` Displays version information.

```
% abstract -V
Abstract version 5.2.2 03/23/05 15:10 (hostname)
```

Displays subversion information.

```
% abstract -W
sub-version ICOA5.2.51.14
```

When reporting problems, provide the information returned by the `-W` argument to customer support.

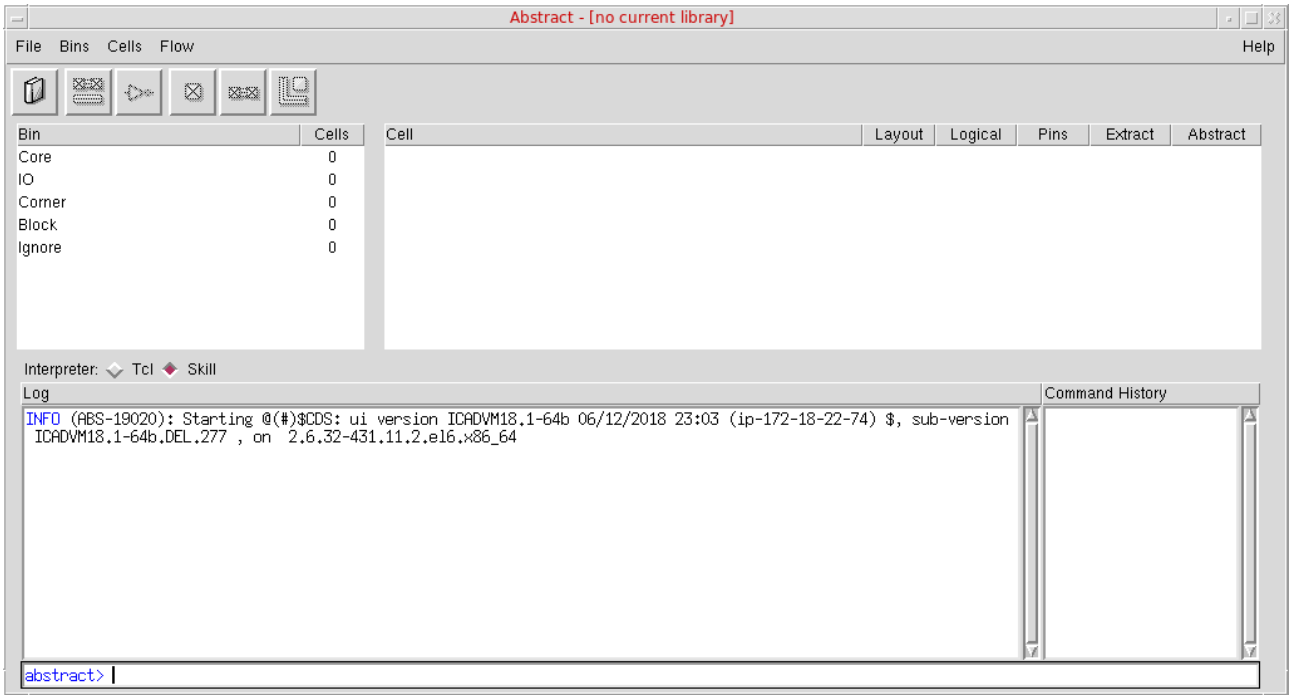
`-dbType` Displays the name of the database used.

```
% abstract -dbType
OPENACCESS
```

`-cdslib path` Specifies the relative or complete path to a cds.lib file. Abstract Generator reads the cds.lib to read the list of libraries to be loaded.

`-help` Displays the list of arguments for the abstract command.

When you type the `abstract` command at the command line prompt, the Abstract Generator form is displayed.



© 2023 Cadence Design Systems, Inc.

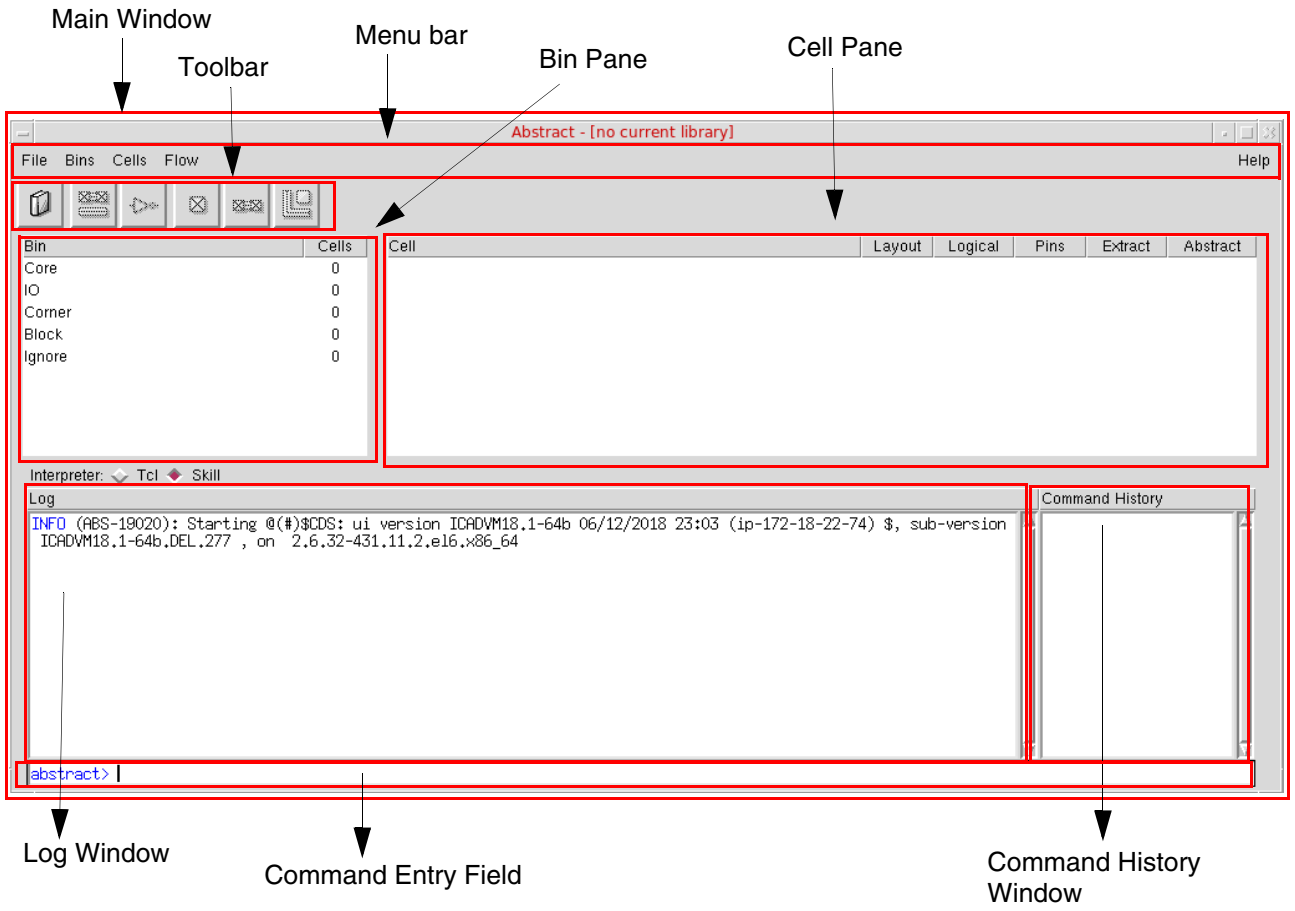
Related Topics

Inspect the Results of Generated Abstracts

© 2023 Cadence Design Systems, Inc.

Standalone Abstract Generator Interface

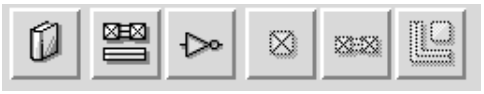
The standalone Abstract Generator interface includes the following components to help you accomplish a wide range of tasks.



The following table describes the main components of the form.

UI Component	Description
Main Window	When you start Abstract Generator for the first time, the main window appears. The application title bar contains the name of the application and the name of the current library if one has been set. If no library has been opened, the title bar of the application displays *no current library* next to the name of the application, as is shown in the figure above.

© 2023 Cadence Design Systems, Inc.

Menu Bar	The menu bar provides access to all Abstract Generator operations. You can also perform some of these operations by quickly using the buttons on the toolbar below the menu bar. Menu commands and toolbar buttons are grayed out when they are not available or do not apply in a particular scenario. For example, until you set a library you cannot import logical or layout information, so these and most of the other commands will be grayed out. Similarly, you cannot use the abstract generation steps until you have selected one or more cells in the Cell Preferences for Abstract Generation
Toolbar	The toolbar is located below the pull-down menus and provides six button shortcuts to the most commonly used functions in Abstract Generator. Moving from left-to-right, the first three buttons are associated with functions in the *File* menu and the last three with functions in the *Flow* menu.  The left-to-right sequence of the buttons implies the basic flow in the abstract generation process, that is, set the current library, import data, and then run the abstract generation steps. You can use these functions in any permitted sequence to achieve more complex flows. Illegal button sequences are prevented by graying-out unavailable functions. ■ *Open Library*: Specifies the library you want to process in Abstract Generator. See Opening a Library in Abstract Generator. ■ *Import GDSII Data*: Creates cellviews from existing layout information in GDSII format. See Importing GDSII Information in Abstract Generator. ■ *Import Logical Data*: Creates cellviews from existing logical information. See Importing Logical Data in Abstract Generator. ■ *Pins*: Lets you view and modify the bin options relevant to the Pins step. ■ *Extract*: Lets you view and modify the bin options relevant to the Extract step. ■ *Abstract*: Lets you view and modify the bin options relevant to the Abstract step.
Bin Pane	The exists on the left of the main application window below the menu bar and toolbar. The *Bin* pane displays the system bins and user bins, if any, and the number of cells in each bin. The number next to each bin name in the *Bin* pane indicates the number of cells in that bin. In the figure above, all the bins are empty. See Bin Preferences for Abstract Generation.

© 2023 Cadence Design Systems, Inc.

Cell Pane	The *Cell* pane is situated on the right of the main window below the menu bar and toolbar. It lists the cells contained in the selected bins and reports the status of those cells. You need to select at least one cell before you can run any of the flow steps or view a status report on a cell. See Cell Preferences for Abstract Generation.
Log Window	The Log window is located in the bottom half of the Abstract Generator main window. The log displays all user interactions with Abstract Generator, including the display of version information on startup, error, warning, and info messages, and TCL/SKILL commands that have been input during a session. Above the Log window, Abstract Generator provides the interface language Interpreter Options, TCL or Skill. Skill is the default and recommended language option. The LOG, ERROR, WARNING, and INFO entries in the Log window are color coded to make it easier to locate serious problems. The LOG and INFO entries are highlighted in BLUE, WARNING messages in ORANGE, and ERROR messages in RED. Each ERROR, WARNING, and INFO message also has a unique message identifier, such as (ABS-246). In this identifier, “ABS” represents “Abstract Generator”, and “246” represents the unique number for identifying the message. All ERROR and WARNING message identifiers are highlighted in clickable BLUE hyperlinks. Clicking the identifier invokes the Doc Assistant in your browser window that displays extended description and possible workaround for the particular ERROR or WARNING message you clicked on.
Command Entry Field	The TCL/SKILL command entry field is at the bottom of the *Log* window and extends up to under the *Command History* window. In this field, at the `abstract>` prompt, you can manually enter a specific command function, for example `absSelectAllBins`, that tells Abstract Generator what operation to perform. The *Command History* window provides other quicker methods of entering command information in this field. See Abstract Generator SKILL Functions.

© 2023 Cadence Design Systems, Inc.

Command History Window

The *Command History* window provides a list of all the TCL/SKILL commands that have been entered in the command entry field.

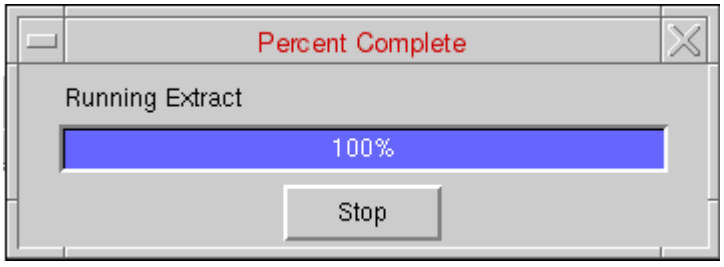
This window helps you save time in re-typing a command. Double-click to select a command in this window, then place your cursor over the Command Entry Field, and click the middle mouse button to copy the command in the command entry field.

Alternatively, you can also scroll through the commands listed in the *Command History* window by using the cursor keys on the keyboard, with whatever command you have highlighted appearing in the command entry field ready for use.

Percent Complete

The *Percent Complete* message box lets you monitor the progress of the current step.

The label on the *Percent Complete* message box tells you what operation is currently being tracked, for example *Running Extract* as in the figure below, while the bar and percentage give you a visual indication of the progress.



Use the *Stop* button in the *Percent Complete* message box to stop the current operation. Abstract Generator prompts you to confirm that you stop the operation.

Depending on the step that is running, Abstract Generator might not be able to stop the operation immediately. In this case, it will proceed until the next safe point before stopping. You can ascertain the current state of your library from the status indicators in the *Cell* pane, and from the log file saved in the run directory.

When you run any abstract generation flow step, Abstract Generator lets you monitor the progress of the step through a message box.

Abstract Generator lets you copy and paste text between GUI fields and tables. To copy text from a GUI field, click-and-drag to select the required text. To paste the text in its new location, click the middle-mouse button in the target field or table cell.

© 2023 Cadence Design Systems, Inc.

To copy text from a table cell, click in the table cell containing the text to be copied. The text for the entire cell is attached to the clipboard until another GUI element is selected. To paste the text in its new location, click the middle-mouse button in the target field or table cell.

Partial text selection for copy and paste is available only in non-table GUI fields.

Related Topics

Custom Fonts in Standalone Abstract Generator Interface

Customized Appearance of the Standalone Abstract Generator Interface

Customization of the Abstract Generation Flow

Standalone Abstract Generator Interface

© 2023 Cadence Design Systems, Inc.

Customization of the Abstract Generator Interface

You can customize the fonts and appearance of the standalone Abstract Generator interface by editing the `.Xdefaults` file. This section explains how to apply the customized changes and demonstrates how to customize the Abstract Generator flow steps.

After you have added the font or the color scheme specification to your `.Xdefaults` file, merge the changes into the X resource database (XRDB) using one of the following commands:

- This command overwrites the previously applied changes.

  ```
  xrdb < <path of the .Xdefaults file>/.Xdefaults
  ```

- This command applies over the previously applied change, even if the setting has been removed from the `.Xdefaults` file.

  ```
  xrdb -merge <path of the .Xdefaults file>/.Xdefaults
  ```

Queries to the XRDB return only the values present when the `.Xdefaults` file was last read in. However, the `.Xdefaults` file is usually read in at login on most UNIX systems.

Custom Fonts in Standalone Abstract Generator Interface

The following default fonts are used by Abstract Generator:

```
*Abstract*font: Helvetica -12
*Abstract*Text*font: Fixed -14
*Abstract*Entry*font: Fixed -13
*AbstractBubbleHelp*font: Helvetica -12
```

The sequence is significant. `*Abstract*font` is the most general rule, so it must be specified before the rules for text and entry widgets so that they can override the general value. If `*Abstract*font` is placed below either one of these, then it will override the values specified by the text and entry rules. The sequence has no effect on the bubble help rule because it uses a separate class, `AbstractBubbleHelp`, to store its resources.

To change all the fonts used in Abstract Generator, add the following line to your `.Xdefaults` file:

```
*Abstract*font: <font specifier>
```

If a font is incorrectly specified, Abstract Generator will not initialize correctly.

The two recommended ways of specifying fonts are described below.

- Standard X Logical Font Descriptor (XLFD): This is used by all X applications and takes the following format:

© 2023 Cadence Design Systems, Inc.

```
-foundry-family-weight-slant-setwidth-addstyle-pixel-point-resx-resy-spacing-
width-charset-encoding
```

For example,

```
-*-Lucidatypewriter-Medium-R-*-*-*-120-*-*-*-*-iso8859-1
```

The individual options are each separated by a hyphen (`-`). An asterisk (`*`) indicates no value for that field.

You can obtain this string using the `xfontsel` utility in a shell window. To use `xfontsel`, use the drop-down menus to select the font required, then click the *select* button to copy the specification to the clipboard.

- TCL font descriptors: The TCL font descriptor requires only the font family, size, and any modifiers (bold, italic, and so on).

  For example,

  ```
  <font family> <size> [<descriptor>+]
  ```

  The sequence is significant; the size must come before the descriptor list.

  Possible descriptor values are

  ```
  normal bold roman italic underline overstrike
  ```

  These values have standard meanings and might not apply for all fonts. For example, fixed-width fonts might not often have an italic variants.

  Examples,

  - `Helvetica -12`
  - `{Times New Roman} -14 bold italic`

    Use braces to enclose a font family that features a space. The hyphen prefixing the font size is required.

To specify different fonts for different GUI widgets, add the following lines to your `.Xdefaults` file:

Widget	Description
Entry widgets	For single-line entry boxes, typically used for entering numbers in the Bin Options form, use `*Abstract*Entry*font: <font specifier>` See Setting Bin Options in Abstract Generator.
Text widgets	For multiple-line text input boxes, used for almost all other textual input in Abstract Generator, use `*Abstract*Text*font: <font specifier>`

© 2023 Cadence Design Systems, Inc.

Bubble Help

For bubble help text, use

```
*AbstractBubbleHelp*font: <font specifier>
```


© 2023 Cadence Design Systems, Inc.

Customized Appearance of the Standalone Abstract Generator Interface

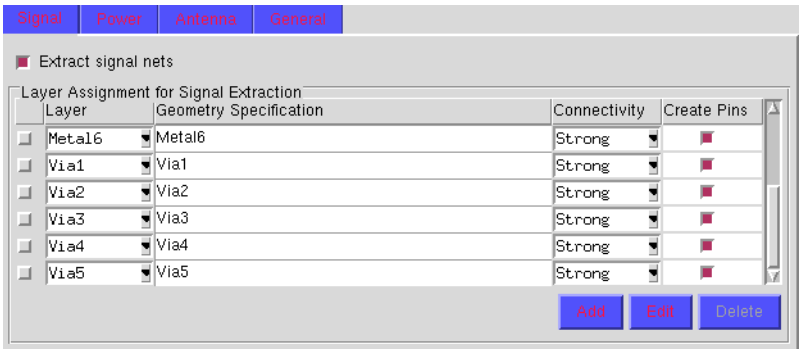
You can specify the color scheme for Abstract Generator UI items and controls, such as tables and buttons, in the `.Xdefaults` file.

The following defaults are used by Abstract Generator:

```
*Abstract*foreground: black
*Abstract*background: gray52
*Abstract*Button.background: gray52
*Abstract*Table.foreground: black
*Abstract*Table.background: medium slate blue
*Abstract*Text.foreground: black
*Abstract*Entry.foreground: black
*Abstract*Label.foreground: black
*Abstract*Label.background: gray52
```

This table describe options to specify the color scheme for the following:

UI Option	Types of Options

Buttons	Use the following options for customizing the colors of buttons and tabs:
	■ This foreground option applies the color setting to the foreground (text) of all buttons and tabs.
	`*Abstract*Button.foreground: <color>`
	■ This background option applies the color setting to the background of all buttons and tabs.
	`*Abstract*Button.background: <color>`
	In the figure below, the text (foreground) on the two active buttons and the four tabs is set to `red` and the button and tab background is set to `blue`.
	 

© 2023 Cadence Design Systems, Inc.

Tables

Use the following options for setting the color scheme for tables:

- This option applies the color setting to the foreground (text) of the selected row(s).

  ```
  *Abstract*Table.foreground: <color>
  ```

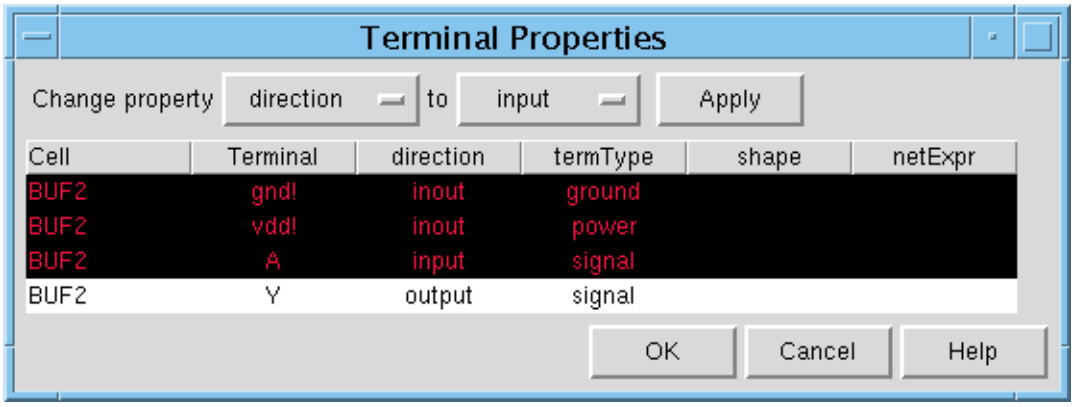
- This option applies the color setting to the background of the selected row(s).

  ```
  *Abstract*Table.background: <color>
  ```

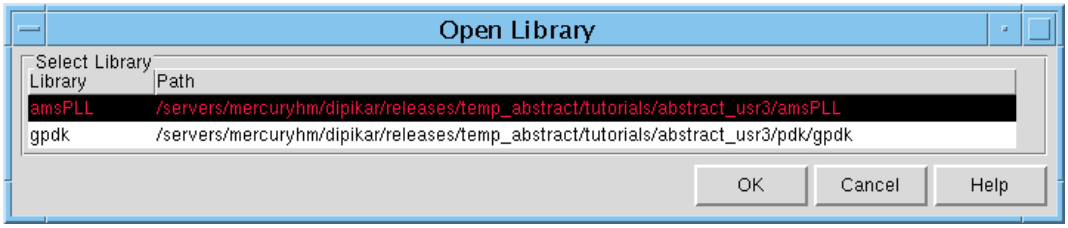
Important

To view the applied color effect, you must select a row.

The figure below illustrates the `black` background and `red` foreground table setting. The applied color effect is showing only for the selected rows.

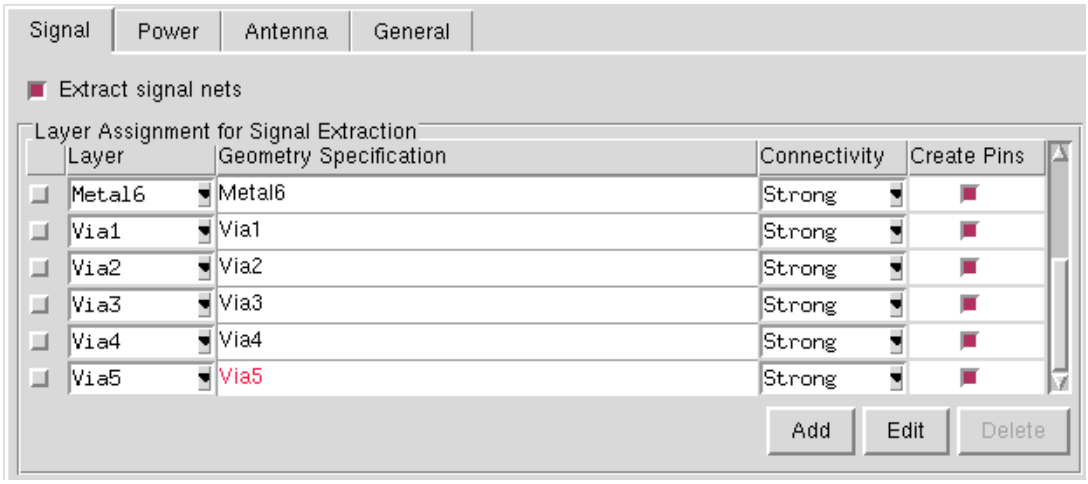


Another example of the same table color setting is provided in the figure below, which captures the *Open Library* form.



© 2023 Cadence Design Systems, Inc.

The *Geometry Specification* column in the *Layer Assignment* tables is also considered to comprise of rows of a table. The text at the current position of the cursor assumes the applied table foreground color. An example is shown in the figure below in which the table foreground color is set to `red` and the cursor is in the last row in the *Geometry Specification* column.



Text Entries

This option applies the color setting to non-numeric text entries and fields expecting non-numeric user text inputs.

```
*Abstract*Text.foreground: <color>
```

In the figure below, the text color has been set to `blue`.

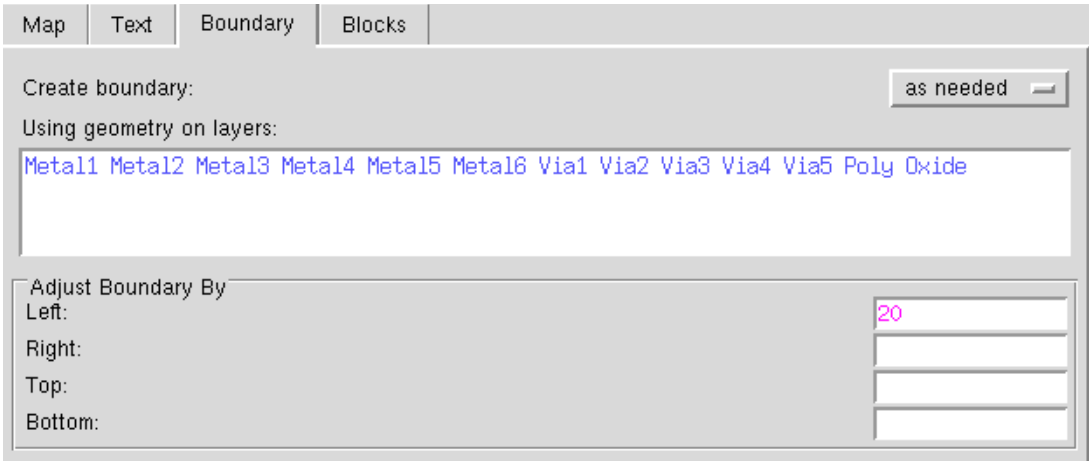
© 2023 Cadence Design Systems, Inc.

Entry Boxes

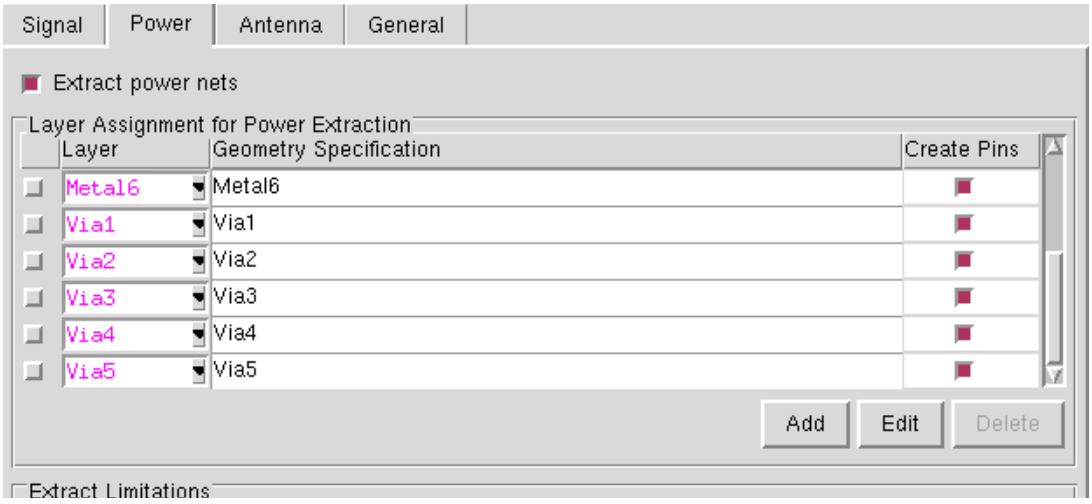
This option applies the color setting to drop-down lists (such as the drop-down lists in the *Layer* column in the *Layer Assignment* tables) and combo boxes, fields with numeric values, and fields expecting numeric inputs.

```
*Abstract*Entry.foreground: <color>
```

The following figure shows the effect of setting the entry box color to `magenta`. The setting for text entry (`blue`) has been retained from the previous example to illustrate the comparison between text entries and entry boxes.



The figure below shows the effect of the same entry box color setting (`magenta`) on drop-down lists.



© 2023 Cadence Design Systems, Inc.

Labels

Use the following options for setting the color of the foreground and background of labels:

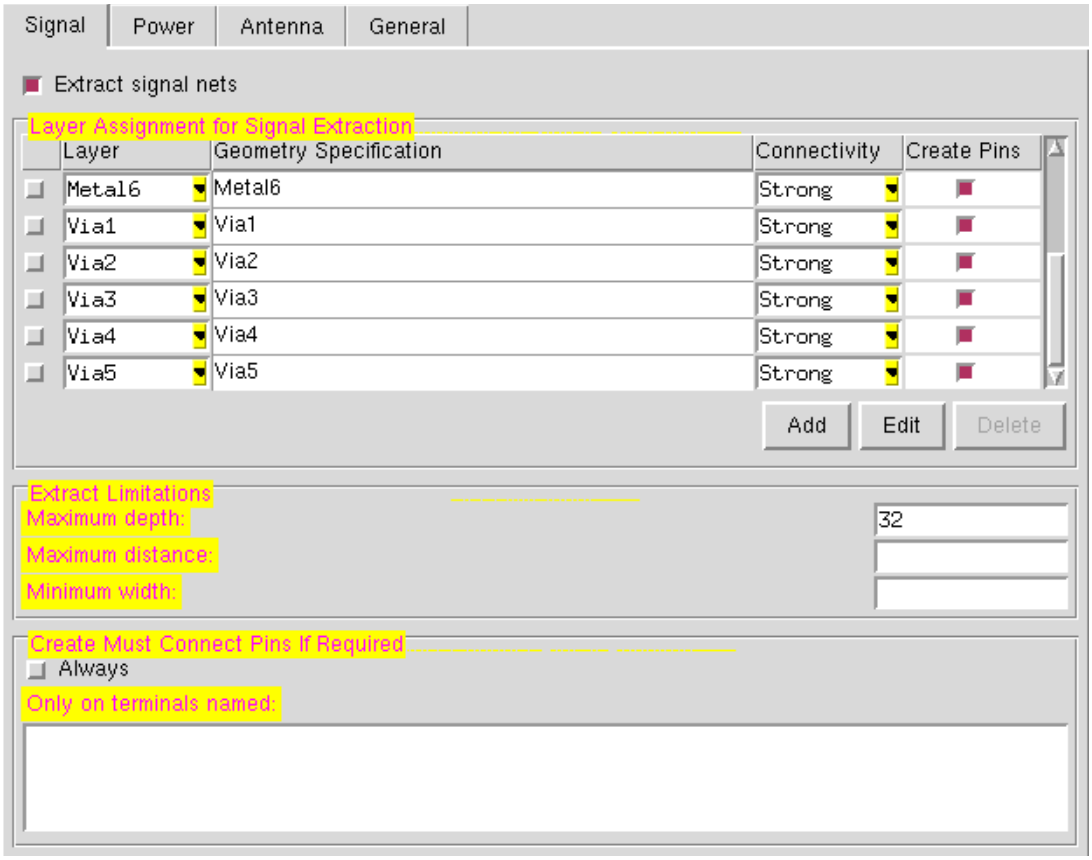
- This option applies the color setting to the foreground of all labels. It overrides the `*Abstract*foreground` setting wherever applicable.

  ```
  *Abstract*Label.foreground: <color>
  ```

- This option applies the color setting to the background of all labels.

  ```
  *Abstract*Label.background: <color>
  ```

In the figure below, the label foreground is set to `magenta` and background to `yellow`.



© 2023 Cadence Design Systems, Inc.

GUI

Use the following options for setting the color of the foreground and background of the GUI:

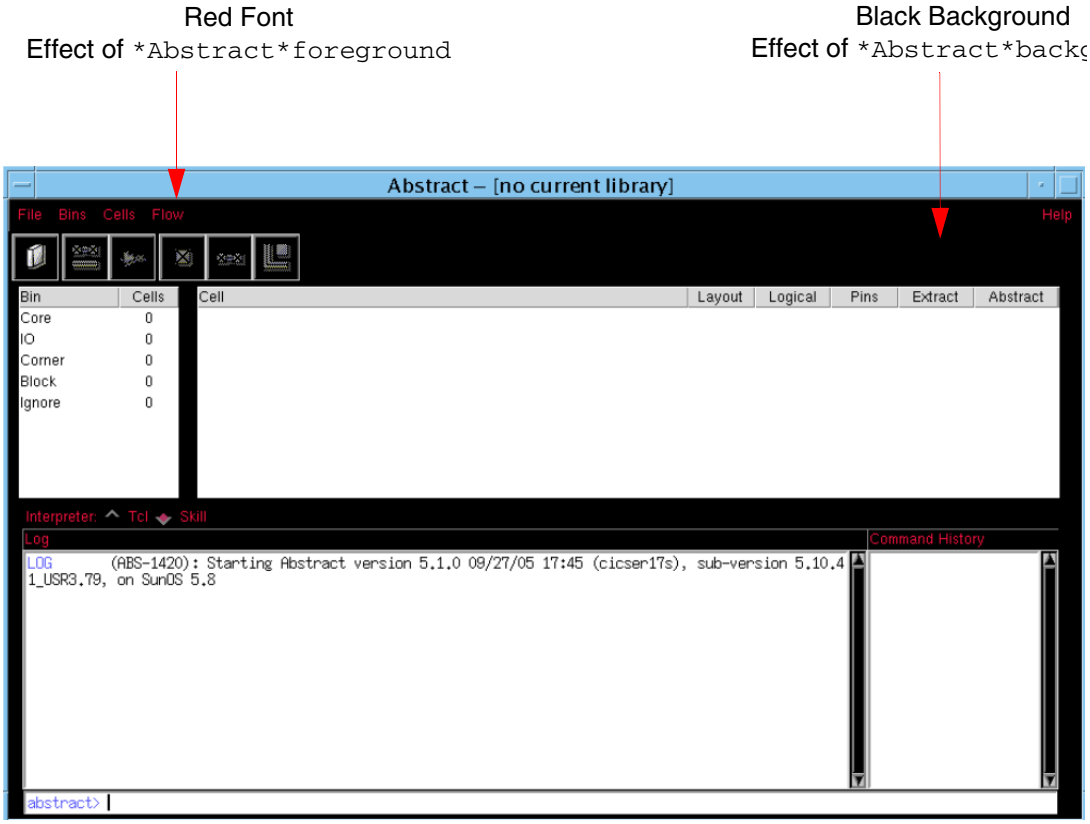
- This foreground option applies the color setting mainly to the GUI text and all active options.

  ```
  *Abstract*foreground: <color>
  ```

- This foreground option applies the color setting to the GUI background.

  ```
  *Abstract*background: <color>
  ```

In the figure below, the GUI foreground is set to `red` and the background to `black`.



After you have added the font or the color scheme specification to your `.Xdefaults` file, merge the changes into the X resource database (XRDB) using one of the following commands:

- This command will overwrite the previously applied changes.

  ```
  xrdb < <path of the .Xdefaults file>/.Xdefaults
  ```


© 2023 Cadence Design Systems, Inc.

- This command will apply over the previously applied change, even if the setting has been removed from the .Xdefaults file.

  ```
  xrdb -merge <path of the .Xdefaults file>/.Xdefaults
  ```

Queries to the XRDB return only the values present when the `.Xdefaults` file was last read. However, the `.Xdefaults` file is usually read at login on most UNIX systems.

Related Topics

Customization of the Abstract Generation Flow

© 2023 Cadence Design Systems, Inc.

Customization of the Abstract Generation Flow

Tasks that you otherwise cannot perform directly by using the available Abstract Generator options can be done using an advanced methodology. This methodology is based on the concept of calling custom code, also called a hook, during the abstract generation flow. This section briefly describes the predefined hooks and how to customize them.

Abstract generator provides predefined hooks, which are automatically called before and after running of a flow step (Pin, Extract, or Abstract). There are therefore, two predefined hooks for every flow step. Following are the predefined hooks in Abstract Generator:

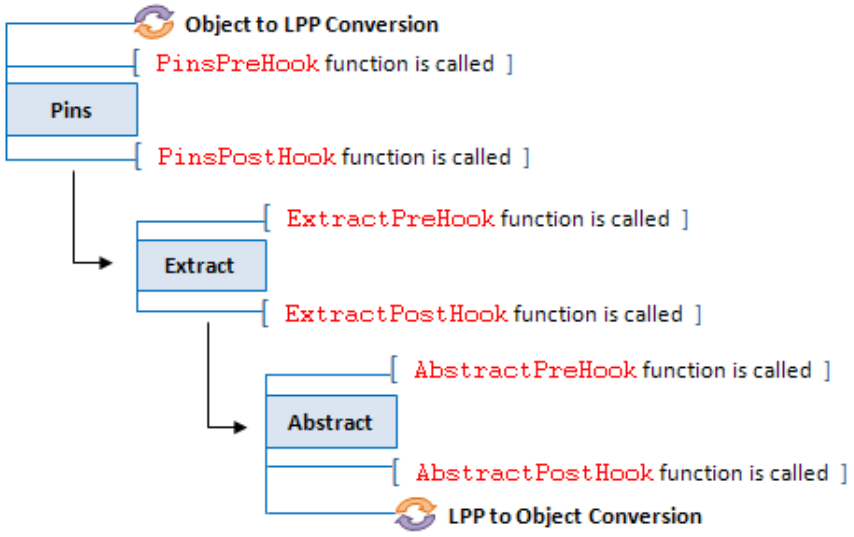
- `PinsPreHook`: This function is called prior to running the Pins step.
- `PinsPostHook`: This function is called after running the Pins step.
- `ExtractPreHook`: This function is called prior to running the Extract step.
- `ExtractPostHook`: This function is called after running the Extract step.
- `AbstractPreHook`: This function is called prior to running the Abstract step.
- `AbstractPostHook`: This function is called after running the Abstract step.
- `ExitHook`: This function is run even if `AbstractPostHook` is not specified, when the tool is closed.

Before the `PinsPreHook` function is called, the pin and blockage objects in the abstract are converted into corresponding pin and blockage shapes.

After running the `AbstractPostHook` function, these shapes are converted into the corresponding objects.

© 2023 Cadence Design Systems, Inc.

The following diagram depicts the sequence of running the predefined hooks:



These hooks are empty by default, and you can redefine them to run a specific task every time a flow step runs.

These predefined hooks exist both in SKILL and TCL. Both SKILL and TCL hooks, empty or redefined, are called for each flow step irrespective of the interpreter selected in the GUI. This ensures consistent results regardless of the interpreter chosen for abstract generation.

The SKILL hooks are called prior to the TCL hooks.

You can modify the definition of the predefined hooks. Ensure to retain the name of the predefined hook while redefining it.

The return value of a hook is not used.

The hooks for the Abstract Generator flow steps require the following arguments:

```
t_binName t_libName t_cellName t_viewName
```

Abstract Generator passes appropriate values for these arguments at run time. The `ExitHook` function is not associated with any specific Abstract Generator step, and therefore does not require any argument.

Example of a Customized AbstractPostHook Function

Consider a scenario in which you need to copy the original pin shape from a layout to the pin purpose in the abstract view. This task cannot be directly performed using any of the available

© 2023 Cadence Design Systems, Inc.

Abstract Generator options. You can, however, use the `AbstractPostHook` function to define this functionality so that it runs automatically after each run of the Abstract step.

Type the following SKILL `AbstractPostHook` code at the `abstract` prompt.

```
procedure(AbstractPostHook(binName libName cellName abstractViewName)
;; This function restores the original pin shapes from a layout in the
;; final abstract generated by Abstract Generator
let((cvL cvA fig layoutViewName)
   layoutViewName = absGet("ViewLayout")
   ;; Opening the layout view
   cvL = dbOpenCellViewByType(libName cellName layoutViewName)
   ;; Opening the abstract view
   cvA = dbOpenCellViewByType(libName cellName abstractViewName "" "a")
   when( cvL && cvA
        ;; Corresponding to each terminal in the layout view, looking for
        ;; terminals in the abstract view
        foreach( term cvL~>terminals
            termA = dbFindTermByName(cvA term~>name)
            ;; Copying pin figures and creating pin shapes corresponding
            ;; to each terminal located in the abstract view
            foreach( pin term~>pins
                fig = dbCopyFig(pin~>fig cvA)
                dbCreatePin(termA~>net fig)
            )
        )
    )
))
```

By default, `AbstractPostHook` option runs before the pin shapes and blockage shapes present in an abstract are translated to the corresponding pin and blockage objects. Consider that there are some shapes on the Metal1 layer in the layout view. When creating a detailed blockage to cover these Metal1 layer shapes, before the post hook runs, these blockage shapes will be created on (Metal1 boundary). To refer to these shapes, you have to refer to the shapes on (Metal1 boundary) LPP. After running post-hook, Abstract Generator converts the Metal1 blockage shapes to Metal1 blockage objects. Thereafter, the blockages can be accessed using the corresponding object ID.

Use the `runAbstractHookPostOat` option to specify whether `AbstractPostHook` needs to be called before or after conversion of pin and blockage shapes into corresponding pin and blockage objects. By default, the `runAbstractHookPostOat` option is set to false. So, `AbstractPostHook` is called before the conversion.

© 2023 Cadence Design Systems, Inc.

When set to true, `AbstractPostHook` is called after the conversion.

```
absSetOption("runAbstractHookPostOat" "false")
```

Customization of the ExitHook Function

Consider a scenario in which you need to run a customized code while exiting Abstract Generator. This task cannot be directly performed using any of the available Abstract Generator options. You can, however, use the `ExitHook` function to define this functionality so that it runs automatically when you close the Abstract Generator window.

Type the following SKILL code at the `abstract` prompt.

```
absSkillMode()
procedure(
ExitHook()
printPort = outfile( "./output.txt" )
println("Exiting Abstract Generator." printPort))
; ; ExitHook() is a procedure that is called when the program exits.
```

When the above procedure is defined, the tool creates a new file named `output.txt` and writes a message to the file indicating that the program is aborted.

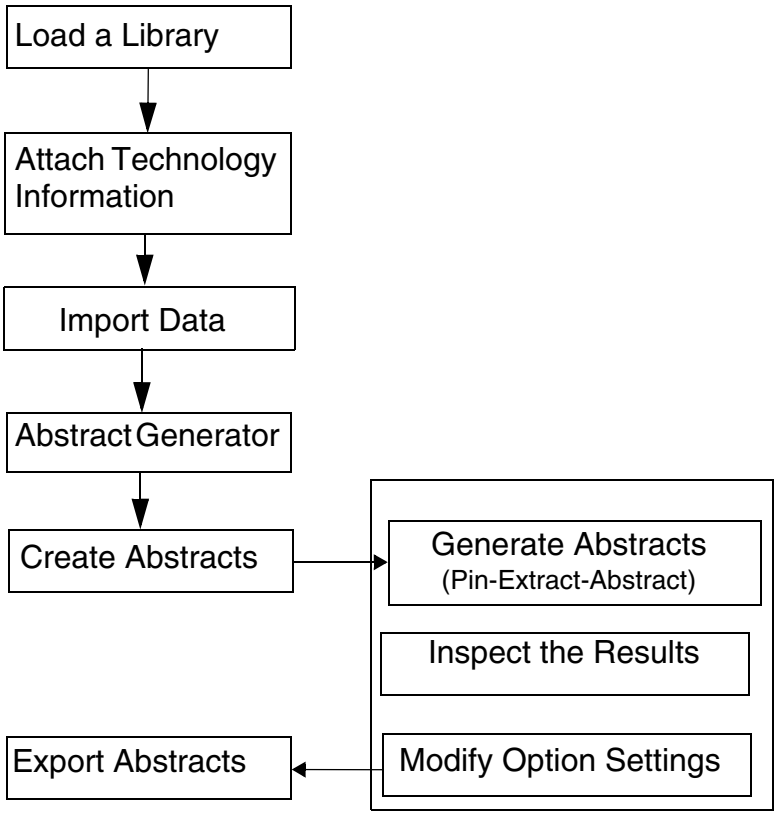
Related Topics

Database Access Functions

© 2023 Cadence Design Systems, Inc.

Abstract Generation Flow

This topic outlines the basic steps required to start generating abstracts for cells or libraries. The following image depicts a flow for generating abstracts in a design.



Before beginning with the abstract generation process, you need to perform the following operations:

1. Open a library.
2. Attach technology information.
3. Import layout information.
4. Import logical information.

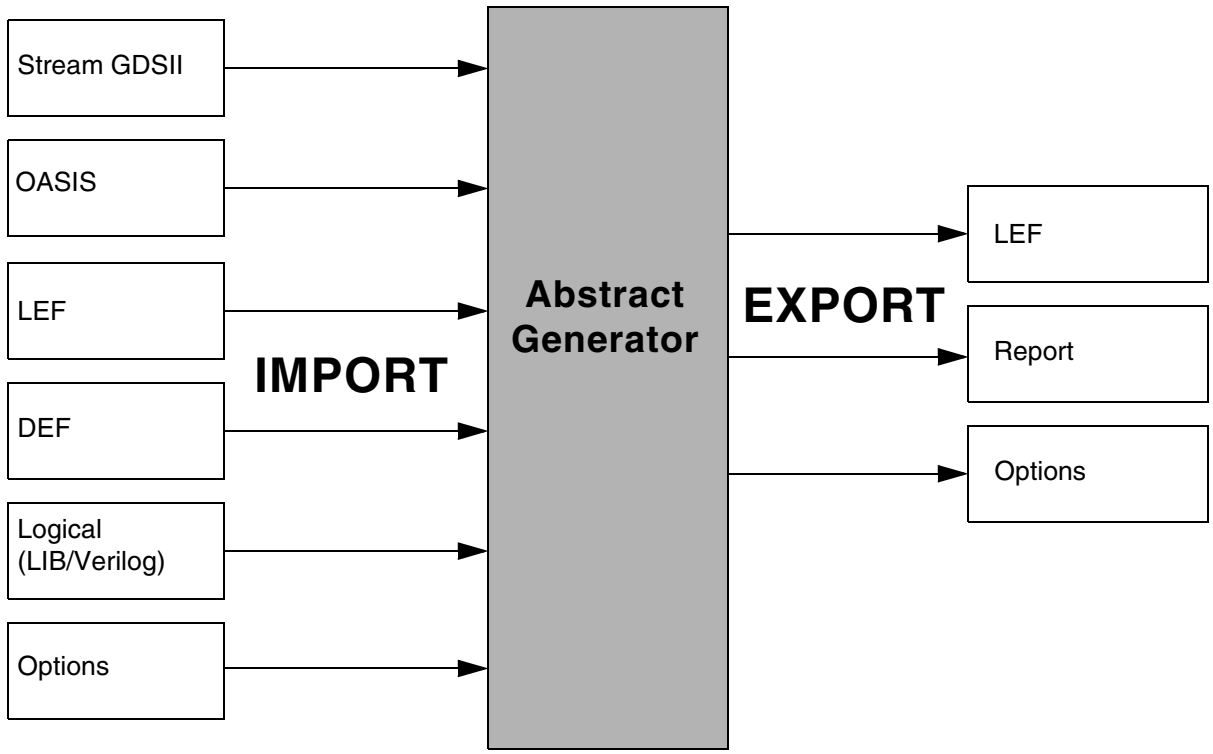
Not all of these procedures are compulsory; see individual procedure for details.

© 2023 Cadence Design Systems, Inc.

Import Data in Abstract Generator

The selected design library may already contain either layout or logical views of cells or views created by Abstract Generator in a previous session. If it does not, you need to import physical layout or logical data. Therefore, after specifying the cell library and attaching the necessary technology data, import information about the cells for which you want to create abstracts. You can import various types of data, such as LEF, DEF, GDSII, and OASIS.

The figure below shows the types of layout and logical data that can be imported into Abstract Generator and the format of data that is exported.



You can import physical layout information, typically represented in the GDSII, OASIS, LEF, or DEF formats. Physical layout information comprises hierarchical shape databases that describe the polygons, paths, and text that make up each individual cell. This data is translated and a layout view for each cell is created in the OpenAccess database.

You can also import logical information, which is represented in Verilog or Liberty format (LIB) formats. This type of information includes data on which pins should be created and the terminal directions assigned. By default, Verilog has input and output terminals defined, whereas LIB has more number of terminals defined including tristate, clock, power, and ground.

Note: You can use the Terminal Properties form to change these settings.

© 2023 Cadence Design Systems, Inc.

Abstract Generator creates logical views in the current library directory. If there is already a layout view present for any given cell, Abstract Generator does not create a new cell but adds the new logical view to any other views already present for the cell. If logical information is imported after running the Pins Step, Abstract Generator confirms that the layout and logical views match before extracting terminal properties from the logical view. See the import options in the File Menu.

Generate Abstracts

Each of the three main flow steps – Pins, Extract, and Abstract have their own set of options that control the way in which any cell is processed. You can make your initial option settings either before you start generating abstracts or when you run any of the individual steps. You can use the Bin Options form and Running Step Abstract form for option settings

You can access the Bin Options form by using the *Bins - Options* menu command. In this form, you can view and modify all options associated with the entire abstract generation flow.

Whenever you run any flow step, Abstract Generator opens the Running Step Abstract form. This form allows you to modify only the options that are relevant to the steps about to be run.

When you are satisfied with the options settings, use the four flow steps to generate abstracts for the selected cells. You can run the steps either one at a time or all at once for any or all of the cells.

Step	Description
Pins	In the Pins step, Abstract Generator creates a place-and-route boundary for the cell and the starting pin shapes for each of the nets to be extracted. It then matches the pins created against those described in any logical view present and appends the appropriate pin direction.
Extract	In the Extract step, Abstract Generator derives which shapes are connected to which nets by tracing the connectivity from the *pin* purpose shapes created during the Pins step. The tool creates a shape with purpose *net* in the top level of the extract view, and for each such shape creates a pin on the appropriate net. The overlap boundary is also calculated if required. Finally, Abstract Generator uses the antenna options to create library process antenna information for custom blocks and standard cells. The resulting antenna model helps mitigate the problem of gate damage caused by chemical and mechanical polishing.

© 2023 Cadence Design Systems, Inc.

Step	Description
Abstract	In the Abstract step, Abstract Generator adjusts the pin shapes created during the Extract step to create the final shapes required by place-and-route tools. It then fractures these pin shapes into rectangles. Next, Abstract Generator applies a layer blockage model selected by the user to create the final blockage geometry in the abstract. The blockage geometry is then ally fractured into rectangles. It then removes from the abstract all layers other than those with purpose *pin*, *blockage*, or *boundary* and deletes the instance hierarchy. At this stage, all the required geometry is at the top level of the abstract.

Inspect the Results of Generated Abstracts

After you have generated abstracts for your sample of selected cells, you must evaluate whether the results are satisfactory. Abstract Generator provides various features to help you verify your abstracts. You can inspect the results using the following interfaces:

Field	Description
Cell Pane	The first source of result evaluation comes from the *Cell Pane* in the main window. Abstract Generator uses color-coded symbols in the *Cell* pane to indicate the result of a particular abstract generation flow step. A color-coded symbol against a cell corresponding to a particular step indicates if the step completed without problems or whether warnings or errors were generated during the step. The *Cell* pane area also indicates when a particular view becomes invalid, for example through a change in an option setting.

© 2023 Cadence Design Systems, Inc.

Layout	If you want to see a detailed graphical representation of any view, you can use Virtuoso Layout Suite XL. You can use this software to examine the pin and blockage geometry generated, the sizings and spacings applied, and to make minor edits. To launch the Virtuoso Layout Suite XL, select *Cells – Edit* and select a view.
Report File and Log File	You can use the information files generated by Abstract Generator to examine potential problems with abstracts. All the messages issued by Abstract Generator are stored in a log file (`abstract.log`) in the run directory. If multiple users are working in the same cell directory, the output is written to the file `abstract.log.1` up to the file `abstract.log.9`. For more structured information, you can export a report file categorizing the cells in the library according to various criteria. You can also view online information about any cell in your library.

Export Abstracts

When you finish generating abstract cellviews, you can export the data in the form of LEF abstracts for use in place-and-route tools.

The *File – Export – LEF* command provides the functionality to translate cell abstracts into LEF, which can be used as input to place-and-route tools.

The *File – Export – Options* command lets you save bin options settings for reuse in a later session. This saves all the current option settings to an `.abstract.options` file in your current working directory.

Abstract Generator writes the following system files in the library directory:

- `.abstract.options`: Stores Abstract Generator option settings for the library. This file is available only in the SKILL format.
- `.abstract.cellstatus`: Stores the Abstract Generator status for all cells in the library. For example, a cellview might have the status `exist`, `exists with errors`, `exists with warnings`, and so on.

Related Topics

Exporting Options File in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Exporting LEF in Abstract Generator

Inspect the Results of Generated Abstracts

Cell Preferences for Abstract Generation

Extract Step in Standalone Abstract Generation

Abstract Step in Standalone Abstract Generation

Pins Step in Standalone Abstract Generation

File Menu in Standalone Abstract Generator

© 2023 Cadence Design Systems, Inc.

Generating Abstracts

Whether you are generating abstracts for an entire library or for a few hierarchical blocks, the approach is typically the same. You can initially focus on a small subset of cells or blocks, establish option settings for this subset, and then process the remaining cells or blocks in a single run.

To generate abstracts:

1. Choose *File – Library – Open* to view the Open Library form.

   This form lists the libraries that are read in from the `cds.lib` file located in the run directory on startup. You can either open an existing design library or create a new one.

2. Select the library you want to process. The library path tells Abstract Generator where to find existing information about the library and where to store the intermediate and final views created during abstract generation.

3. You can attach a technology library to the design library you have selected to process. If you attach a technology library to the design library or if there is a technology library associated with the design library, Abstract Generator uses the existing technology library information, such as the `.abstract.options` file.

4. Import various types of data, such as LEF, DEF, GDSII, and OASIS if the selected design library does not contain layout or logical views of cells or views created by Abstract Generator in a previous session.

5. Choose *Select* and *Sort* functions of Abstract Generator to locate the cells you are interested in, and then use the manual and automatic *Distribute* functions to move these cells into the appropriate system or user bins. These commands are available on the *Cells* menu.

   Abstract Generator uses the concept of Bins to distribute the library cells into a number of mutually exclusive sets. When you import cell information into Abstract Generator, it is by default assigned to the *Core* bin. This might be sufficient for certain libraries, but there will be cases where you require a more sophisticated distribution before you start processing your cells.

6. Identify the cells with which you want to experiment, and begin to Generate Abstracts for these cells. Each of the three main flow steps – Pins, Extract, and Abstract have their own set of options that control the way in which any cell is processed. You can make your initial option settings either before you start generating abstracts or when you run any of the individual steps.

7. Inspect the results in the generated abstracts using the *Cell* pane, layout, and Report File and Log File.

© 2023 Cadence Design Systems, Inc.

8. Modify the option settings as required and rerun the relevant steps to generate a new abstract after you assess the abstracts.

   When you change options related to a particular step, the view created by that step is invalidated along with all “downstream” views present for that cell.

9. Choose *Cells – Revalidate Selected* to re-validate invalid views. When you are satisfied with the results of your tests and the options you want to use for each bin, you can process the remaining library cells either bin-by-bin, step-by-step, or all at a time.

10. Choose *File – Export – LEF* to translate cell abstracts into LEF, which can be used as input to place-and-route tools.

Related Topics

Cell Preferences for Abstract Generation

Inspect the Results of Generated Abstracts

Options and Status Files in Abstract Generator

Attaching Technology Information to a Library in Abstract Generator

Opening a Library in Abstract Generator

Creating a Library in Abstract Generator

© 2023 Cadence Design Systems, Inc.

File Menu in Standalone Abstract Generator

You can use the commands in the *File* menu to specify all setup information, ranging from defining technology information and specifying a library through importing physical and logical data for the cells and blocks to be processed, for Abstract Generator to process a library. The commands for specifying this information are also available on the toolbar.

You can open and create a library, attach technology information to a library in Abstract Generator. The commands for importing GDSII information, OASIS data, LEF and DEF are also available on the *File* menu. Other commands in the *File* menu let you export LEF descriptions of abstracts, generate a statistics report for the current library, and save the option settings of the current session. You can also replay a previous session.

Related Topics

Creating a Library in Abstract Generator

Importing GDSII Information in Abstract Generator

Importing DEF in Abstract Generator

Importing Options File in Abstract Generator

Exporting LEF in Abstract Generator

Specifying Global Options in General Options form

© 2023 Cadence Design Systems, Inc.

Creating a Library in Abstract Generator

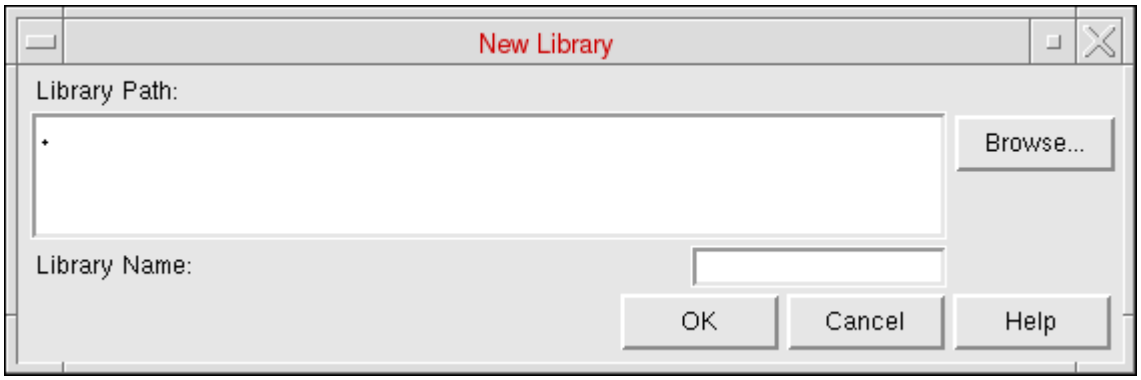
If you use *File – Library – New* command to create a new library, Abstract Generator automatically attaches the Cadence default technology file shipped with the software, `cdsDefTechLib`, to the library. The default technology file is unlikely to contain enough information to satisfy the internal technology checker of Abstract Generator, so you must always replace or supplement the technology information before you begin abstract generation.

There are two ways of doing this using the Abstract Generator GUI:

- Select *File – Library – Attach Technology* to attach an existing technology file to be used with your new library. During abstract generation, the tool might need to update your technology information; for example, the tool might update the technology file with information about sites and layer-purpose pairs. You must ensure that an attached technology file has permissions that allow it to be updated by Abstract Generator, otherwise you will need to add the missing information manually.
- Use *File – Import – LEF* to import a LEF technology file. Abstract Generator automatically converts the information in the LEF file into appropriate technology information for OpenAccess.

The New Library form allows you to create a new library in the specified directory. You can create a library at any time during the abstract generation process.

1. Choose *File – Library – New*. It displays the New Library window.



2. In *Library Path*, specify a valid directory path to where the new library should be created. The default value is the current directory, that is, “.”.
3. Click *Browse* to display the Browse New Library Directory form that can be used to locate the required directory location for the new library.

© 2023 Cadence Design Systems, Inc.

If no path is specified, a warning message is displayed requesting that you enter a valid directory path.

4. In *Library Name*, specify a valid library name. The library name must not contain any spaces. You also cannot specify the name of an existing library that is already specified in the `cds.lib` file of the current directory.
5. Click *OK*.

A new library will be created at the location specified (a new library directory will be created if one does not already exist, with the same library directory name as the name of the library). The library is added to the `cds.lib` file and a copy of `cdsDefTechLib` is added to the library. The default `cdsDefTechLib` technology library is copied into the new library to allow the import of technology LEF to create a technology file. When you are importing LEF in Abstract Generator, you can chose to select the *Overwrite* option which lets you copy over the current technology.

Related Topics

New Library Window

Importing LEF in Abstract Generator

© 2023 Cadence Design Systems, Inc.

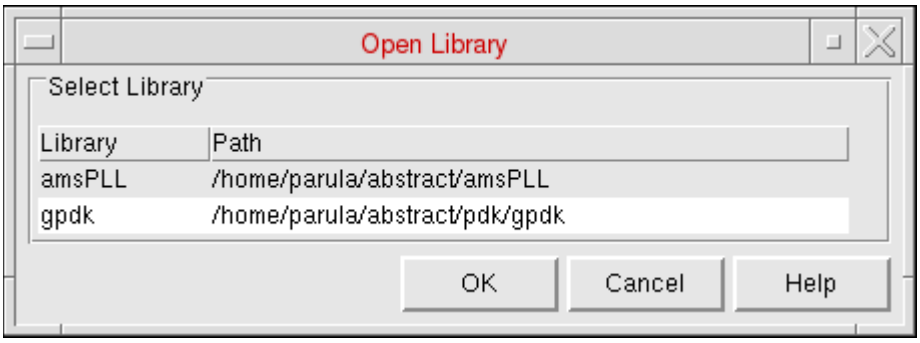
Opening a Library in Abstract Generator

The Open Library window lists the libraries and their associated paths that are currently registered in Abstract Generator. These are read in from the `cds.lib` file located in the run directory and are listed in an alphabetical order. The `cds.lib` file defines the libraries that tools read to identify the libraries they can use. The file maps user library names to physical directory paths. Usually, one `cds.lib` file, which might reference other `cds.lib` files, determines the libraries that are available to your design tool.

1. You can specify the library you want to process in Abstract Generator by selecting *File – Library – Open* or by clicking the *Library* button in the toolbar.

   If there is only one library registered, clicking the *Library* button on the toolbar automatically opens that library for the current session.

   Note: The *File – Library – Open* menu option is grayed out if there are no libraries listed in the `cds.lib` file.



2. Select the library you want to open and click *OK*.

All the views created by Abstract Generator, including the final abstracts, are stored in this library. The option settings from a previous Abstract Generator session are stored in your current working directory in a `.abstract.options` file. When you reopen the library, Abstract Generator reads in this and restores the bin options to the settings that existed when the library was last saved.

Related Topics

Open Library Window

Cadence Library Definition File

Troubleshoot Library Issues

© 2023 Cadence Design Systems, Inc.

Troubleshoot Library Issues

You can face following issues while attaching a technology information to a library in Abstract Generator.

ABS-10517

The *name* library has been interrupted while loading. *value* cells out of *value* have been successfully loaded.

To resolve this issue:

➡ Load the cell library again without interrupting Abstract Generator.

ABS-10526

Failed to attach the '%s' technology library to the '%s' name library. Ensure the following:

1. Use the `absAttachTechLib` command after setting a library. You can do this only in `nogui` or `replay` modes.
2. You have permission to write in the design library, specifically the prop.xx file, which contains property information regarding the attached technology library.
3. The design library exists and that it is not locked.
4. The technology library you are trying to attach exists.

Related Topics

Opening a Library in Abstract Generator

© 2023 Cadence Design Systems, Inc.

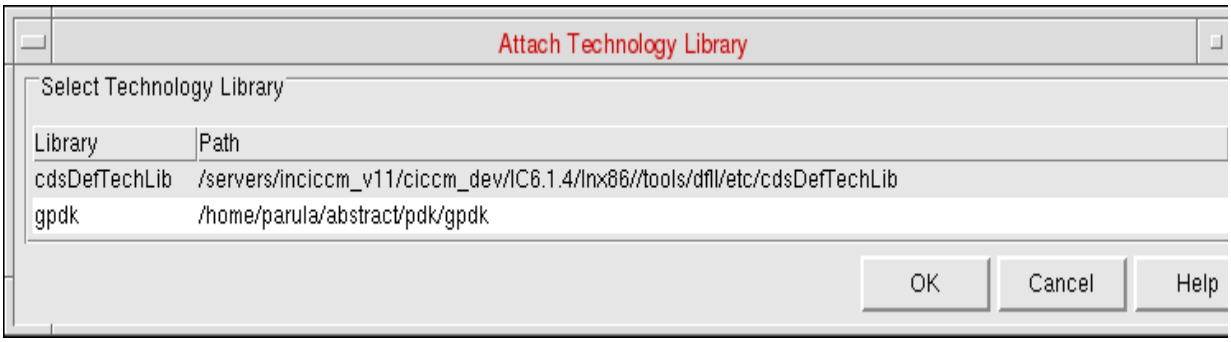
Attaching Technology Information to a Library in Abstract Generator

The Attach Technology Library form allows you to attach and associate existing technology library information to the current cell library. This form lists all the libraries in the `cds.lib` file that have their own `techfile.cds` file contained in the library itself. The library list displayed also includes `cdsDefTechLib`, which is the default technology library. You cannot import a technology LEF file if the default technology file, `cdsDefTechLib`, is already attached to a cell library. If you attempt to do this, an error message will display.

If you are importing an existing library into Abstract Generator and the library has its own technology file (`tech.db`), Abstract Generator uses that file as the main source of technology information. If the existing library has a different technology library attached, Abstract Generator uses that instead. If your library has an attached technology file, Cadence recommends that you move `tech.db` file out of the library directory before you start generating abstracts so that possible conflicts can be avoided.

1. Choose *File – Library – Attach Technology.*

   The Attach Technology Library form is displayed. The *Attach Library* menu option is available only if a cell library is already open.



2. Select a library and click *OK*.

If you choose to attach a library, Abstract Generator attempts to combine this library to the current library and load it into the software. If you attempt to attach the same technology file that is already associated with the current library, you will be given the opportunity to reattach that technology library.

Related Topics

Attach Technology Library Form

© 2023 Cadence Design Systems, Inc.

Troubleshoot Technology Issues

This section lists the common problems found in technology files in the Virtuoso Studio design environment and provides information about how to fix these to enable Abstract Generator to run.

ABS-10034

A list of errors is displayed in the technology file. There are a number critical checks performed by Abstract Generator on start-up, among them a validity check on the technology data associated with a design. If any of these checks fails, Abstract Generator issues an appropriate error message. Use the Technology File Manager to resolve these errors before proceeding.

ABS-10042

Technology initialization failed because the name layer with the value layer number and the name layer with the value layer number have the same name. The same layer name cannot be associated with two layer numbers. Each layer has an associated layer name and is assigned a unique layer number in `techLayers` in the `layerDefinitions` section of the technology file. Rectify the error in the technology file and then try again.

ABS-10044

Failed to perform `antennaSideArea` calculations because the name layer does not have a thickness value specified. To resolve this issue:

1. Specify the thickness value in the *antenna* subsection of the *foundry* constraint group of the technology file.
2. To perform antenna side area factor or ratio calculations, set a `thickness` value for the layer in the `antenna` subsection of `foundry` constraint group.

ABS-10050

Failed to create the name, purpose layer-purpose pair in the technology file. Ensure that you have permissions to write to the technology file and then try again. To do this:

1. Specify the layer-purpose pair information in the `techPurposes` construct in the `layerDefinitions` section during the abstract generation process.

   If missing, you can append this information to the technology file.

© 2023 Cadence Design Systems, Inc.

2. If write permissions exist on the technology file, Abstract Generator will automatically add the missing layer-purpose pair. However, you would not be able to update the technology file if you do not have the write permission on the file, which is when Abstract Generator issues this message.
3. Change the permission setting of the technology file or contact the person responsible for updating the technology file.
4. Use the Technology File Manager to add the missing layer-purpose definition to the technology file prior to invoking Abstract Generator.

ABS-10053

Abstract Generator has failed to set the required property. If you are unable to isolate the problem, contact your Cadence Customer Support representative for help.

ABS-10054

Failed to initialize Abstract Generator because either the *name* layer is not defined in the *standardViaDefs* subsection of the *viaDefs* section or the layer function has not been identified as *'cut'* in the *functions* subsection of the *layerRules* section. Ensure that the layer is correctly defined in both *standardViaDefs* and *functions* subsection and then try again.

1. You need to define via layers in the `standardViaDefs` section and as ‘cut’ in the `functions` subclass of the `layerRules` section. This error will be generated if the via layer is not defined in any one of these.
2. Using the Technology File Manager, check the `standardViaDefs` section and the `functions` subsection in the `layerRules` section in the technology file.
3. Update the section in which the via layer definition is missing.

ABS-10055

Failed to create the *name* site in the *technology library name* technology file. Abstract Generator tried unsuccessfully to create a site in the OpenAccess technology file, `tech.db`. The exception message issued by OpenAccess indicates the reason for the failure. For example, you might not have permission to update the `tech.db` file.

ABS-19006

The routing direction for layer *layerName* has not been specified. Specify the appropriate direction in the *routingDirections* sub-section of *layerRules* section of the technology file.

© 2023 Cadence Design Systems, Inc.

Related Topics

Technology File Requirements for Abstract Generation

Technology File Manager

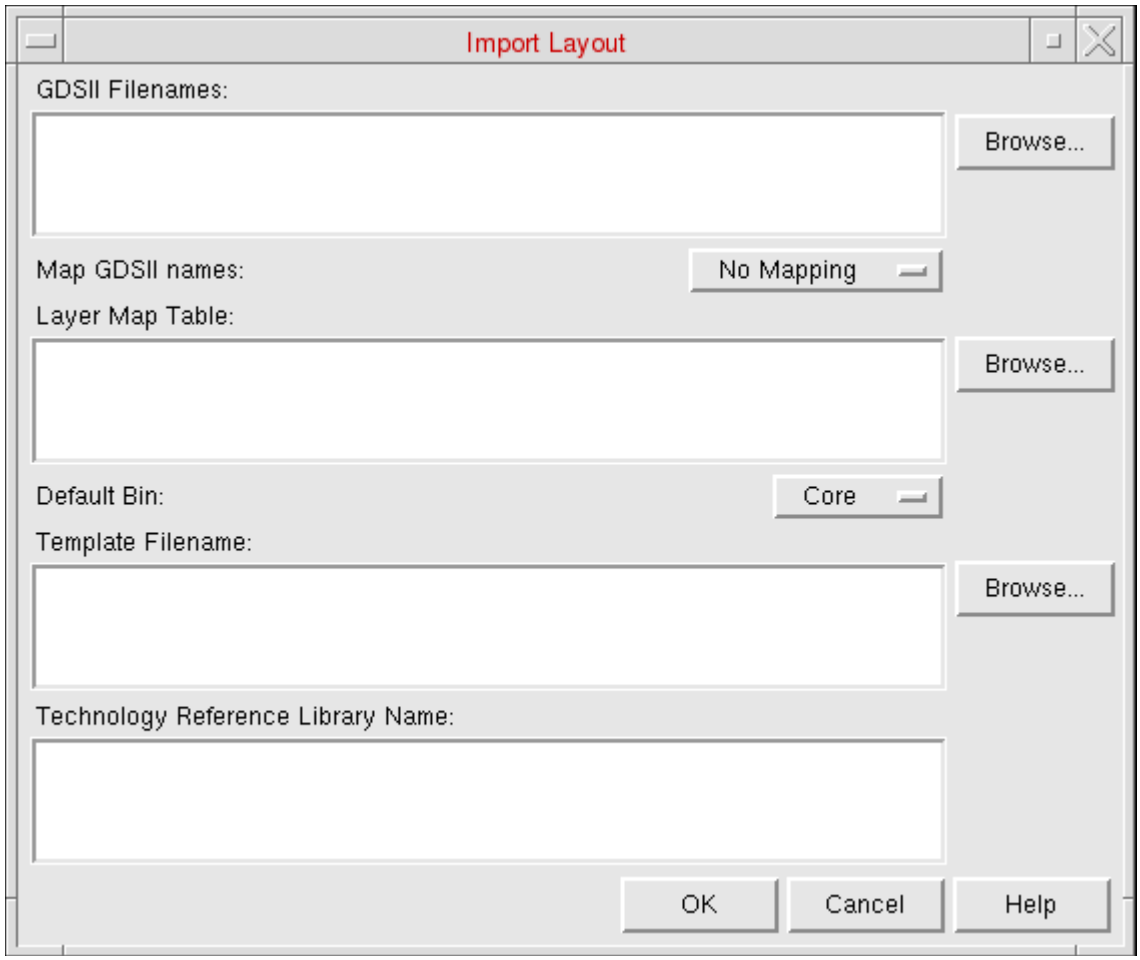
© 2023 Cadence Design Systems, Inc.

Importing GDSII Information in Abstract Generator

The Import Layout form creates cellviews from existing layout information in GDSII format. The OpenAccess version of Abstract Generator uses XStream to import GDSII information.

1. Choose *File – Import – Stream (GDSII)* to start this command or click the *Layout* button in the toolbar.

   This displays the Import Layout form.



2. In *GDSII Filenames*, specify the names of the GDSII files that you want to import. Use the *Browse* button to help you locate the files.
3. In *Map GDSII names*, you can specify whether you want GDSII names mapped to upper case, lower case, or to not be mapped at all.
4. In *Layer Map Table* , specify the name of the layer map file you want to import. Use the *Browse* button to help you locate the file. A layer map file is an ASCII file that maps the

© 2023 Cadence Design Systems, Inc.

layer numbers and data types used in the Stream file to the layer-purpose pairs in the OpenAccess database. See OpenAccess database.

5. In *Default Bin*, you can select the bin into which the cells are imported. This option applies only to any new library cells being imported. Cells imported previously are not moved.

   After importing, if you select the *Default Bin* in the main window, you see the imported cells listed in the right-hand pane. The check mark in the *Layout* column indicates the presence of a layout view. Cells are also checked for valid layers.

6. In *Template Filename*, you can specify a stream template file to be imported into Abstract Generator. Use the *Browse* button to help you locate template files.

7. In *Technology Reference Library Name*, you can specify one or more valid reference technology libraries. The specified referenced technology libraries must be present in the `cds.lib` file and must be readable.

Enabling Coloring While Importing GDSII Information

Use the `ImportGDSIIEnableColoring` to specify whether coloring needs to be enabled while importing GDSII data. By default, the option is set to `"true"`.

```
absSetOption("ImportGDSIIEnableColoring"  "true")
```

For color layouts to be generated, in addition to setting the `ImportGDSIIEnableColoring` option to `"true"`, ensure that layer map file includes mask information.

Related Topics

Import Layout Form

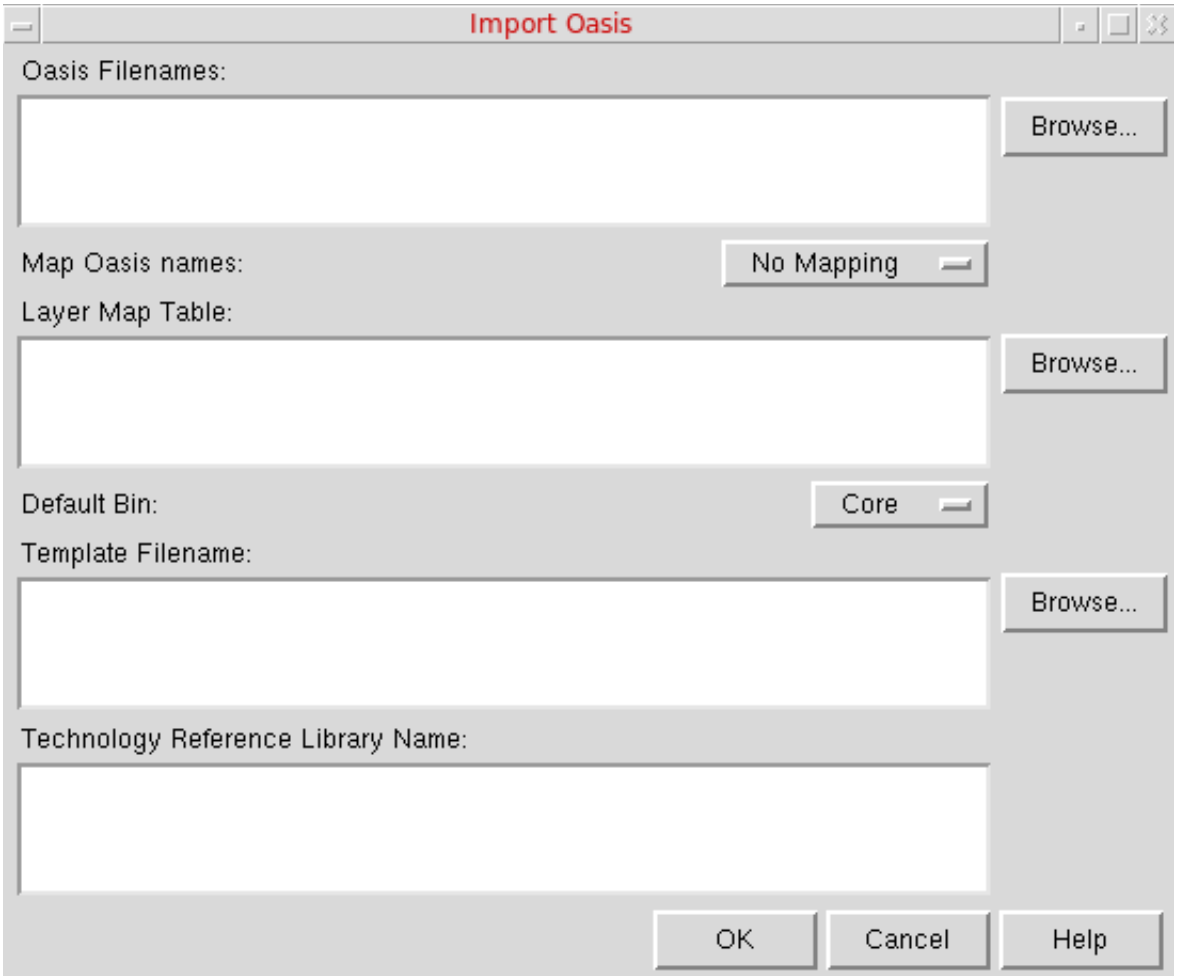
© 2023 Cadence Design Systems, Inc.

Importing OASIS Data in Abstract Generator

The Import Oasis form creates cellviews from existing layout information in OASIS format.

1. Choose *File – Import – Stream (Oasis)* to start this command.

   This displays the Import Oasis form.



2. In *Oasis Filenames*, specify the names of the OASIS files that you want to import. Use the *Browse* button to help you locate the files.
3. In *Map Oasis names*, you can specify whether you want OASIS names mapped to uppercase, lowercase, or not to be mapped at all.
4. In *Layer Map Table*, specify the name of the layer map file you want to import. Use the *Browse* button to help you locate the file. A layer map file is an ASCII file that maps the layer numbers and data types used in the Stream file to the layer-purpose pairs in the OpenAccess database.

© 2023 Cadence Design Systems, Inc.

5. In *Default Bin*, you can select the bin into which the cells are imported. This option applies only to any *new* library cells being imported. Cells imported previously are not moved.

   After importing, if you select the *Default Bin* in the main window, you see the imported cells listed in the right-hand pane. The check mark in the *Layout* column indicates the presence of a layout view. Cells are also checked for valid layers.

6. In *Template Filename*, specify a stream template file to be imported into Abstract Generator. Use the *Browse* button to help you locate template files.

7. Specify one or more valid reference technology libraries in the *Technology Reference Library Name* field. The specified referenced technology libraries must be present in the `cds.lib` file and must be readable. The subsection `techControl`, added to the technology file for ITDB (Incremental Technology Database) support, necessitates the specification of a set of ordered reference libraries to allow inheritance from these libraries.

Related Topics

Import Oasis Form

Design Translation Using XStream Translator

© 2023 Cadence Design Systems, Inc.

Importing LEF in Abstract Generator

The Import LEF form lets you create cellviews from LEF data. This is used when creating blocks from DEF. The LEF data for the leaf cells should be imported, followed by the DEF data for the associated block.

1. Selecting *File – Import – LEF* displays the Import LEF form.

2. The *LEF Filenames* field specifies the LEF files to import. You can type the name manually in the field or *Browse* for existing files with the `.lef` extension in the current directory.
3. The *View* field specifies the cellview to which a LEF macro will be saved. The default view is `abstract` but this can be changed so that the LEF macro is saved to another view, such as the `layout` view.
4. The subsection `techControl`, added to the technology file for incremental technology database support, necessitates the specification of a set of ordered reference libraries to allow inheritance from these libraries.

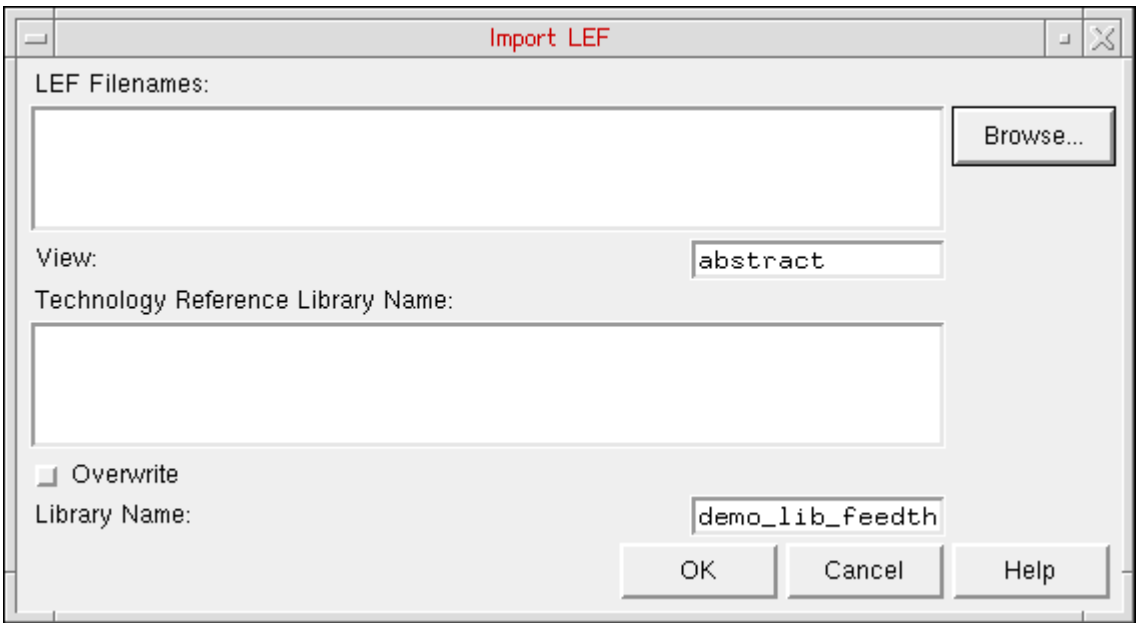
   Abstract Generator allows you to specify one or more valid reference technology libraries in the *Technology Reference Library Name* field. The specified referenced technology libraries must be present in the `cds.lib` file and must be readable.
5. Select *Overwrite* to overwrite the current technology file if the LEF file to be imported contains one. This option is off by default.

   All kinds of LEF files can be imported, ones with both technology and geometry information and also the ones with only technology or only geometry information.

© 2023 Cadence Design Systems, Inc.

Note: You cannot import a technology LEF file if the default technology file, `cdsDefTechLib`, is already attached to a cell library. If you attempt to do this, an error message will display.

6. In *Library Name*, specify the name of the LEF library that needs to be imported.



The library and its associated technology file are loaded into the current abstract session. At this point, the corresponding GDS file, if available, is also read.

When specifying the *Library Name*:

- If a library is already loaded in the current abstract session, then the name of the currently open library is displayed in the *Library Name* field. Either retain the same library name or specify a different value.
- If you have created a new library, then the name of the newly created library is displayed in the *Library Name* field. Either retain the same library name or specify a different value.
- If no library is currently loaded in AG, and user wants to create new library using the Import LEF form, then it is mandatory to specify the library name along with the LEF file name. Else an error message is displayed and the Import LEF operation fails.

7. Click *OK*.

Abstract Generator opens the specified library. So, if you specify a library name other than the currently loaded library, then the newly created library is loaded.

© 2023 Cadence Design Systems, Inc.

Related Topics

Incremental Technology Databases

© 2023 Cadence Design Systems, Inc.

Importing DEF in Abstract Generator

A DEF file contains the design-specific information of a circuit and is a representation of the design at any point during the layout process. The DEF file is in ASCII format. DEF conveys logical design data to and physical design data from P&R tools. Logical design data can include internal connectivity (represented by a net list), grouping information, timing parameters and physical constraints. Physical data includes placement locations and orientations, routing geometry data, and logical design changes for backannotation.

1. Choose *File – Import – DEF*. It displays the Import DEF form.

   The *File – Import – DEF* command imports a net list in DEF format and updates the design with the information.

2. The DEF specifies the DEF files to import. You can type in the name manually or *Browse* for existing files with the `.def` extension in the current directory.
3. In *Default Bin*, select the bin to which DEF data is to be imported.

   Cells will be placed in the *Default Bin* only if they conform to the following criteria:

   - There is a valid layout view.
   - The cell is not a via.
   - The cell is not a site.

   If any of this criteria is not met, the cells will be placed in the *Ignore* bin.
4. Click *OK*.

Related Topics

Import DEF Form

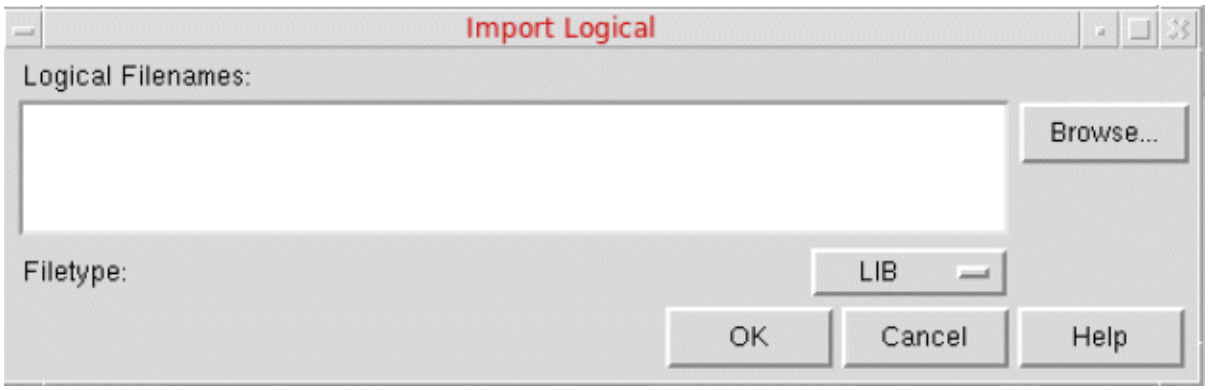
© 2023 Cadence Design Systems, Inc.

Importing Logical Data in Abstract Generator

The Import Logical form creates cellviews from existing logical information, typically represented in Liberty or Verilog® formats.

1. Select *File – Import – Logical* or click the *Logical* button in the toolbar.

   This displays the Import Logical form.



2. In *Logical Filenames*, specify the names of one or more files you want to import, or use the *Browse* button to help locate the file and add it to the selection.
3. You can filter the search by using the *Filetype* list.
4. Choose the required file format, and click *OK* to import the selected file.

Abstract Generator creates cells with logical views in the current library directory. If the cell already exists, Abstract Generator does not create a new cell but adds logical view information to the other views present for the cell.

If you import logical information after you have run the Pins step, Abstract Generator confirms that the layout and logical views match before extracting terminal properties from the logical view. Cells that have only a logical view go to the *Ignore* bin by default.

Important

You cannot import Verilog or Liberty information if you have set the global option to pick up logical information from a `schematic` or `symbol` view.

Related Topics

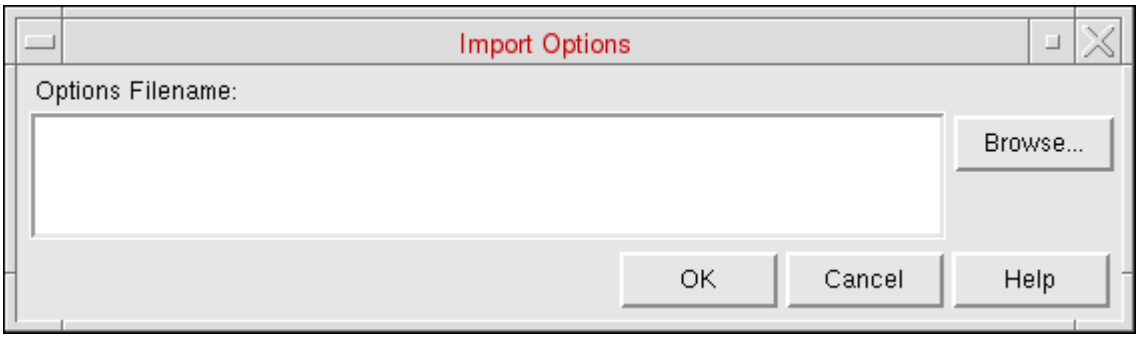
Specifying Global Options in General Options form

© 2023 Cadence Design Systems, Inc.

Importing Options File in Abstract Generator

The Import Options form allows you to load an options file saved in a previous session. You can save the settings of a session and load these settings back when you start a new session using the *Import* command.

1. Choose *File – Import – Options* to display the Import Options form.



2. In the *Options Filename* field, specify the path and name of the file to import, or use *Browse* to look for the file through various folders.
3. Click *OK* to import the options file from the specified location.

Related Topics

Import Options Form

© 2023 Cadence Design Systems, Inc.

Creating an Abstract from a DEF Block

Abstract Generator can be used to generate abstracts and process antenna information for routed blocks.

1. Choose *File – Import – LEF* option to import your LEF file.

   Note: When generating process antenna data for a routed block, you only need standard cell LEF, which has antenna information on the terminals.

   Ensure that the DEF file contains pins for VSS/VDD with no geometry.

2. Run the *Pins* step.

   - In the Extract step of *Power* tab, leave the *Extract power nets* option off.
   - In the Abstract step of *Adjust* tab, select the style of power pins that you want to create, for example, boundary or pin.

   This solution should work irrespective of the standard cells inside the block having feedthru or abutment pins and no matter what style of power pin you select for the block itself.

   If the DEF file does not contain power pin statements for VSS/VDD, add these statements to the DEF file pins section, as shown below:

   ```
   VSS + NET VSS + DIRECTION INOUT + USE GROUND
   VDD + NET VDD + DIRECTION INOUT + USE POWER
   ```

3. Run the Extract step.

   - When calculating the process antenna effect (PAE), it will be necessary to run `do_extraction`. The recommended way of doing this is to leave the *Extract signal nets* option off, select all the appropriate antenna calculation buttons on the *Antenna* tab, and list the polysilicon, contact and diffusion layers in the *Layer Assignment for Extraction* table. The recommended antenna calculation to be performed here is *Calculate hierarchical antenna*, which does not require `DIFF`, `CONT`, or `POLY` layers in the extraction.
   - With this set-up, Abstract Generator will `do_extraction` to generate the PAE data but will not create any new pins. The pin geometry would essentially be the same as defined in the DEF pins section.

4. Run the Abstract step.

   - In the DEF flow, where each pin in the DEF file contained geometry, it is recommended to turn the *Create boundary pins* option off.

© 2023 Cadence Design Systems, Inc.

- For *Power Nets*, if the power pin geometry was defined in the DEF file, it is likely that this is the desired pin geometry in the abstract. In this case, it is recommended that both *Create boundary pins* and *Create ring pins* options be turned off.
- If the DEF file contains no geometric pin shape, you should select the *Create boundary pins* option for *Signal Nets* and *Create boundary pins* or *Create ring pins* option for *Power Nets*.

Related Topics

Import LEF Form

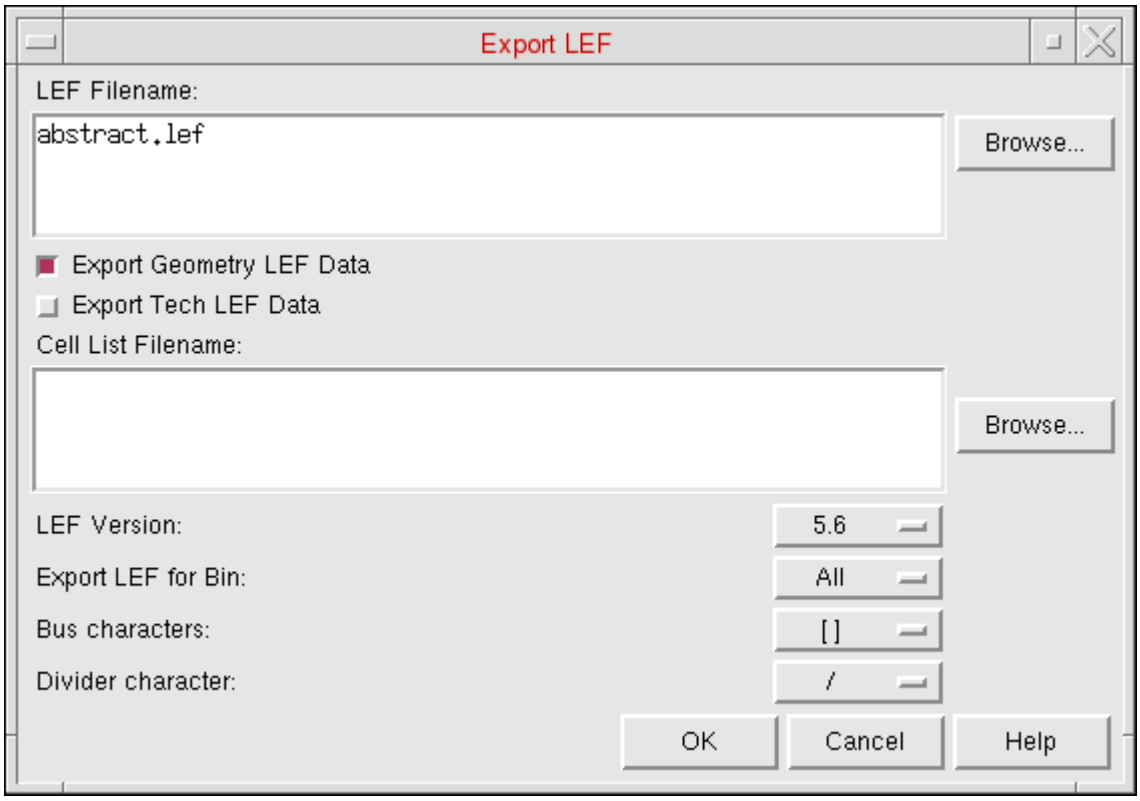
© 2023 Cadence Design Systems, Inc.

Exporting LEF in Abstract Generator

The Export LEF form generates LEF descriptions for cells in one or more bins.

1. Choose *File – Export – LEF*.

   It displays the Export LEF form.



2. In *LEF Filename*, specify a path and name for the LEF file to export or use the *Browse* button to specify a location.
3. Select *Export Geometry LEF Data* and *Export Tech LEF Data* to choose the combination of geometry LEF and technology LEF to be exported. All the LEF information is written to the specified LEF file name.
4. Select *Cell List Filename* to export LEF information for only the specified cells. The list of cells, whose LEF information is to be exported, is specified in a cell list file, with one cell name per line. Specify the name of this cell list file in the field or use *Browse* to specify the file's location. This is an optional field.

   If you specify a cell list file, the *Export LEF for Bin* option gets disabled.
5. In *LEF Version*, select the version of LEF that you want to export.

© 2023 Cadence Design Systems, Inc.

6. In *Export LEF for Bin*, select the bins for which you want to export the LEF data. You can export LEF for all the cells in your library or on a bin-by-bin basis.
7. In *Bus characters* and *Divider character*, select the characters you want to use to represent buses (< > or [ ]) and pseudohierarchy (/ or |), respectively, in your LEF.
8. Click *OK* to export the LEF files.

The resultant LEF file can be imported into the target P&R system.

Including DATABASE MICRONS in the Exported LEF

To set `DATABASE MICRONS` values in the LEF information exported from the CDB Abstract Generator, you must set the `gec3DBUPerUU` value in your CDB technology file.

In OpenAccess, the `DATABASE MICRONS` value is taken directly from the `DBUPerUU` value defined in the technology file.

Related Topics

Export LEF Form

DATABASE MICRONS in the Exported LEF File

© 2023 Cadence Design Systems, Inc.

Troubleshoot Import and Export Issues

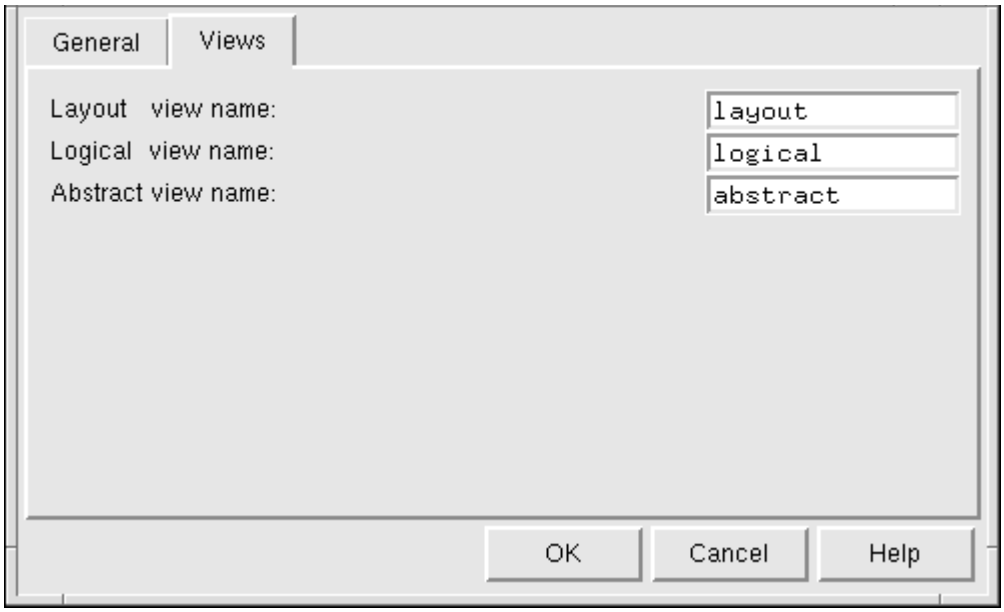
ABS-16001

Cell *name*: The *Import - Logical* command failed to create a cellview. This is because no view name has been specified for importing the logical information (Verilog).

Troubleshooting:

To enable the abstract generator to successfully import the verilog file and create the cellview:

1. Specify a view name in the *Logical View Name* field in the *Views* tab of the *General options* form.



2. Ensure that the view name (logical) is compatible with the type of logical data (verilog) being imported and then try again.
3. Check the verilog file to ensure that its syntax is correct. See Specifying Global Options in General Options form.

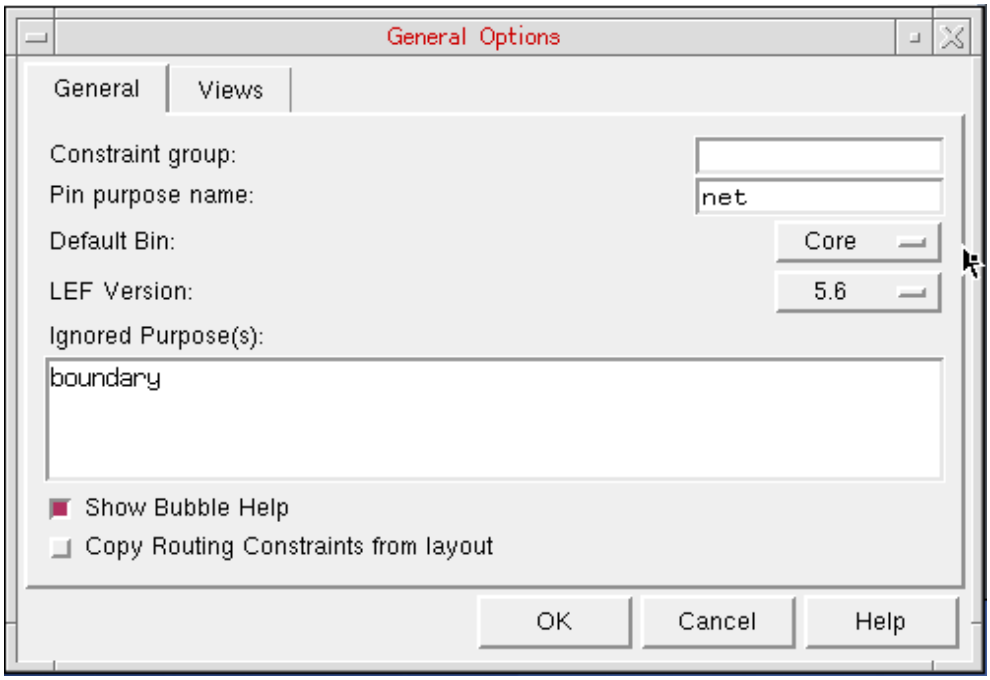
ABS-16002

Failed to import the logical information because the *name* logical view name is invalid. Change the *Logical view name* option to a valid name in the General options form and then try again.

© 2023 Cadence Design Systems, Inc.

Troubleshooting:

In the *File – General options* dialog, you have specified a logical view name which is incompatible with the type of logical file you are trying to import.



For example, the option might be set to "`schematic`" and you are trying to import a Verilog file. Ensure that the global logical file name is compatible with the file type.

ABS-16100

Bin *`name`*: Export LEF failed because there are no valid abstracts or there were failures with the tech LEF or technology file could not be opened.

Troubleshooting: If the problem is related to technology, bin, or cell failures, then it is likely that difficulties were encountered during the `LEFOUT` translation process. Ensure that you have the required permissions for the technology file and then try again.

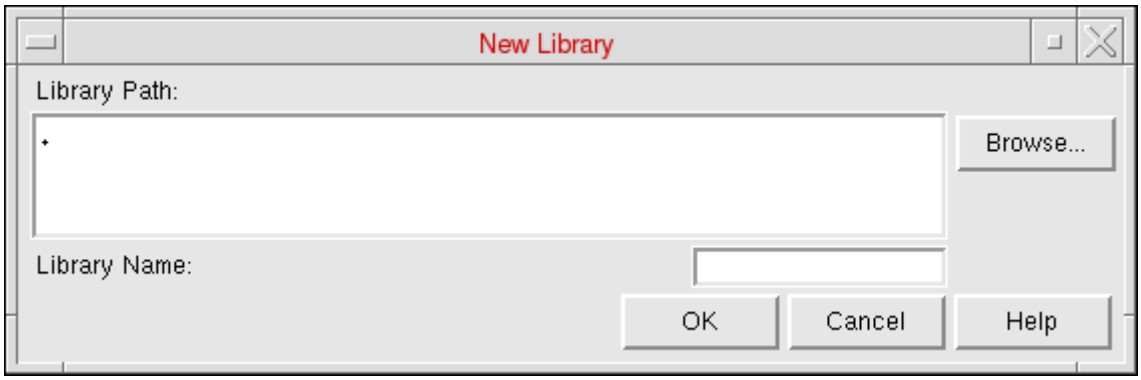
ABS-16005

Failed to import the technology LEF because the current attached technology library is the default *`name`*.

Troubleshooting:

© 2023 Cadence Design Systems, Inc.

You might have created a new library using *File – Library – New* and did not specifically choose to attach a particular technology library (using *File – Library – Attach Technology*) and `cdsDefTechLib` was added by default. Attach a non-default technology library to merge LEF technology information and then try again.



See Creating a Library in Abstract Generator.

ABS-16006

Failed to import the technology LEF because the *display.drf* file is not writable.

Troubleshooting: Ensure that the permissions on the `display.drf` file are set correctly so that the technology LEF can be imported.

ABS-16103

LEFOUT failed. Either no valid abstract views are present in the library or the cells given in the cell list file are not found in the abstract with valid views.

Troubleshooting: Ensure that the cells specified in the cell list file exist in the library.

ABS-16104

LEFOUT failed. File name containing the list of cells is invalid or is not found in the given path.

Troubleshooting: Ensure that the correct path to the cell list file has been specified and that you have read permission to the file.

© 2023 Cadence Design Systems, Inc.

ABS-17000

Failed to import *Name* option because the option has not been specified. Specify the option and then try again.

ABS-17003

The *Name* global option has an invalid value and has been reinitialized from *value* to *value*.

Troubleshooting: When there is a change in the technology file that is being used any invalid global options must be reinitialized.

ABS-17004

The *name* bin's *Name* option has an invalid value and has been reinitialized from *value* to *value*.

Troubleshooting: When there is a change in the technology file that is being used any invalid bin options must be reinitialized.

ABS-17006

The entry in the *Text Manipulation* table for bin *name* which maps '*string*' to '*string*' has been changed to map '*string*' to '*string*' in order that pin names are created with angle brackets rather than square brackets. Re-import the logical view and rerun the Pins step to ensure that the generated abstract views are correct.

Troubleshooting:

Previously, bus characters were represented using square brackets; now they are represented using angle brackets. If Abstract Generator finds any entries that map to square brackets, it changes them as follows.

- Entries mapping `( )` to `[ ]` are deleted.
- Entries mapping `< >` to `[ ]` are changed to map `[ ]` to `< >`.

ABS-17007

Detected an entry in the *Text Manipulation* table for bin *name* which may be mapping angle brackets or parentheses to square brackets (maps '*string*' to '*string*').

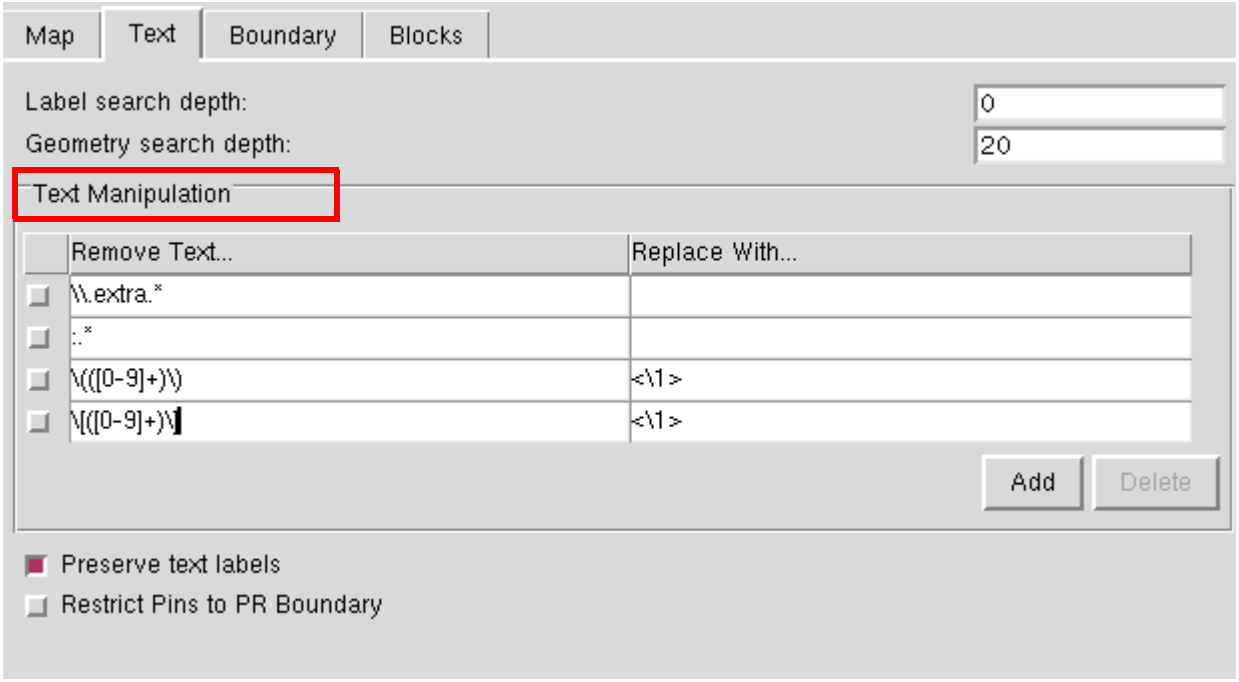
Troubleshooting:

The bus characters are represented internally as angle brackets, rather than square

© 2023 Cadence Design Systems, Inc.

brackets. To resolve the issue:

1. Map the entries in the *Text Manipulation* table to square brackets (e.g., '\\[([0-9]+)\\]' to '<\\1>').



2. Once this option has been set up correctly, re-import the logical view.
3. Run the Pins step in order to ensure the generated abstract views are correct.s

ABS-17008

The global option `ImportLogicalType` has been assigned to an invalid value TLF.

Troubleshooting: The valid values for the `ImportLogicalType` global option are `LIB` and `Verilog`.

Related Topics

Customizing Text Labels for Abstract Generation

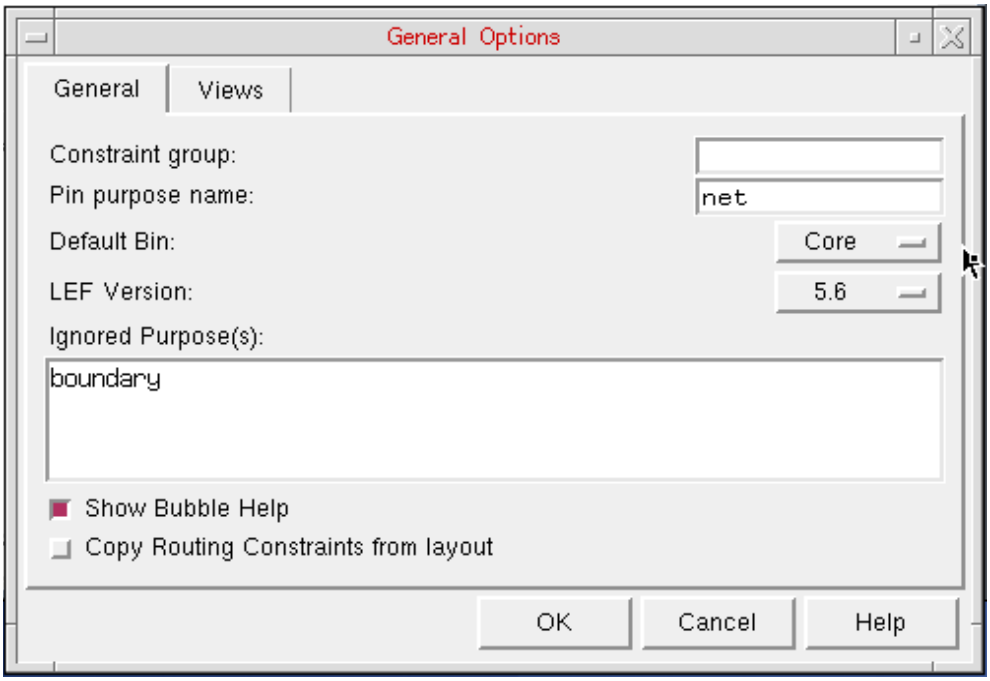
© 2023 Cadence Design Systems, Inc.

Specifying Global Options in General Options form

The General Options form lets you view and change options that apply to the overall Abstract Generator. The options are split into different categories, accessed using the *Views* and *General* tabs in the General Options form.

1. Choose *File – General Options* command.

   The General tab provides the options to set the default bin, LEF version, and the purpose name(s) to be ignored during the flow steps.

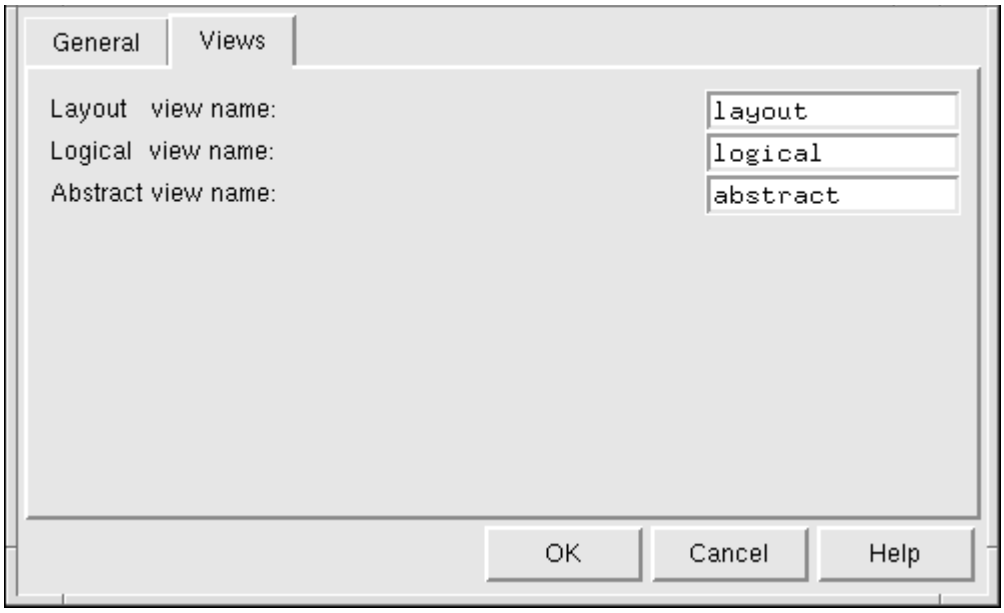


   - Select *Constraint group* to specify the name of the constraint group of the technology database.
   - Select *Pin purpose name* to set the purpose for pins on the final abstracts. The default purpose is `net`.
   - Select *Default Bin* to specify the default bin into which library cells are imported.
   - Select *LEF Version* to set the version of LEF to be used by Abstract Generator. You can select from `5.4`, `5.5`, `5.6`, and `5.7`. The default version of LEF used by Abstract Generator is `5.7` for mature nodes and `5.8` for advanced nodes.
   - Select *Ignored Purpose(s)* to specify the purpose name(s) that you want to be ignored when a layer name without a qualifying purpose name is specified for the Pin, Extract, and Abstract steps.

© 2023 Cadence Design Systems, Inc.

- Select *Show Bubble Help* to display bubble help for various GUI options. It is selected by default. You can turn it off if required.
- Select *Copy Routing Constraints from layout* to enable Abstract Generator to copy the mixed signal routing constraints from the layout view to the abstract view.

2. In the *Views* tab, you can specify the file names that Abstract Generator would use while creating layout, logical, and abstract database files. The *Logical file name* field can also be used to specify an existing logical view (for example, `schematic` or `symbol`) from which logical information is to be picked up.



- Select *Layout View Name* to specify the view name for layout information. For example, if you import GDSII layout data into Abstract Generator, this is the name used for the database files that are generated.
- Select *Logical View Name* to specify the view name for logical information.
- Select *Abstract view name* to specify the name used for the abstract views created by Abstract Generator.

3. Click *OK*.

Related Topics

General Options Form

© 2023 Cadence Design Systems, Inc.

Viewing the Statistics Report in Abstract Generator

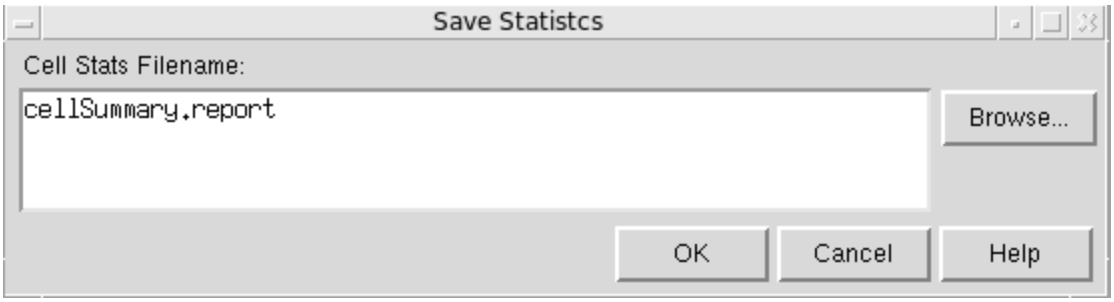
At any point during the Abstract Generator run, use the statistics report to see a report window that displays the abstract generation status and statistics until that point.

1. Choose *File – Statistics Report*.

   A separate statistics report is displayed for each selected cell.

2. The < and > buttons are useful when multiple cells are selected for abstract generation. Use these buttons to navigate between reports for the various cells. A new statistics report is generated for each Abstract Generator run.

3. Click *Save As*. The Save Statistics window is displayed.



4. Specify a different *Cell Stats Filename*. The default filename is `cellSummary.report`.

5. To select an existing file, click *Browse* and select the required file.

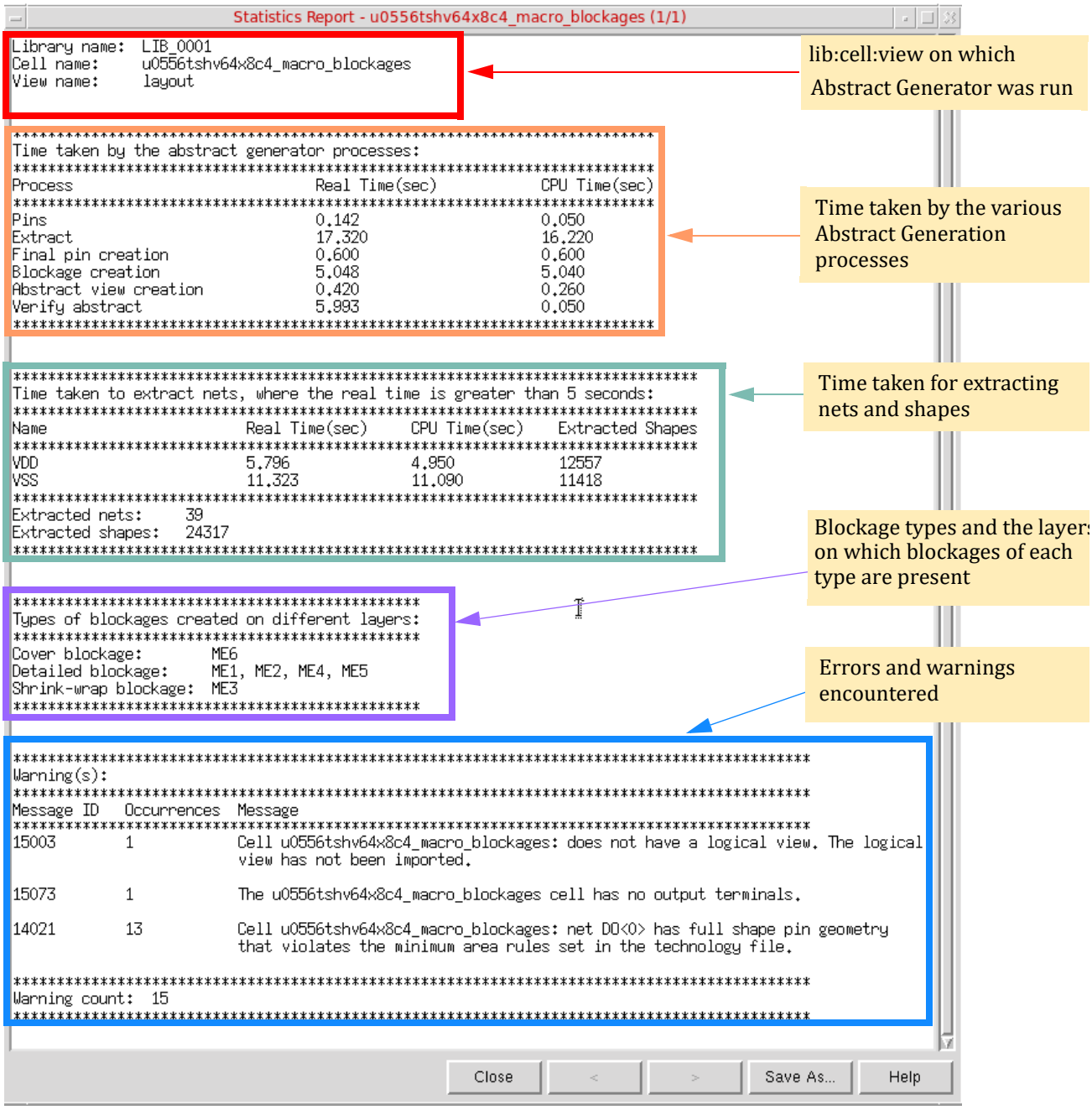
   Note: If a file with the specified name already exists, then a message indicating this is displayed. You can choose to either replace the existing file or save the report with a different filename.

6. Click *OK*.

   The statistics report is stored in the text format.

© 2023 Cadence Design Systems, Inc.

The report provides the following information about the abstract generation run on each cell:



Related Topics

Specifying Global Options in General Options form

© 2023 Cadence Design Systems, Inc.

Bin Preferences for Abstract Generation

Bins are like buckets for placing cells. Each cell in a library is always contained in one bin. Each bin is associated with certain options, which allows the cells in a library to be distributed into mutually exclusive sets for processing. For example, Abstract Generator uses the option settings for `binA` to determine how to treat the cell `cellA` in `binA`.

There are essentially two types of bins, systems bins and user bins.

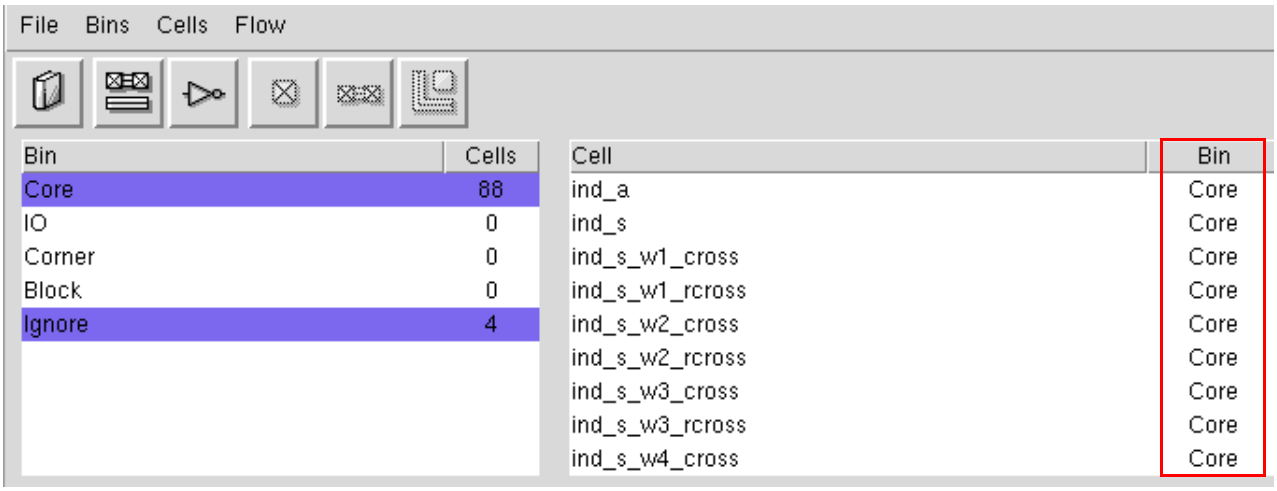
- System Bins: There are five *system bins,* one for each of the four main types of cells — *Core, IO, Corner, Block* and one to store any cells that you do not want to process at a particular time —*Ignore*. Any cells placed in the *Ignore* bin are not deleted but are excluded from the set of cells to be processed.

  Note: You cannot add, rename, or remove system bins.

- User Bins: If you require bins of any given type in addition to the systems bins, you can create user bins. For example, you might want to create additional bins so that you can experiment with different option settings on two or more sample sets of cells. Or you might want to divide the core cells in your library into two separate sets to process some cells using slightly differing option settings and compare the results generated. You can do this up by creating a user bin and specifying its type as *Core*, and then applying option settings to it different from those of the *Core* bin.

  User bins differ from system bins only in that you can delete and rename user bins, while system bins are permanent and cannot be renamed or deleted.

When you select a bin, you see the contents of that bin listed in the *Cell* pane.If you select two or more bins simultaneously, the contents of all the selected bins are displayed in the *Cell* pane. This gives you complete flexibility in defining the set of cells to be processed without compromising on the different options set for the cells.



© 2023 Cadence Design Systems, Inc.

When you select multiple bins, you can use the *Bin* column in the *Cell* pane to recognize the bin to which a cell belongs.

The *Bins* menu contains commands to perform the following tasks:

- Creating a User Bin in Abstract Generator
- Adding a User Bin in Abstract Generator
- Renaming a User Bin in Abstract Generator
- Deleting a User Bin in Abstract Generator
- Setting Bin Options in Abstract Generator

Related Topics

Cell Preferences for Abstract Generation

Standalone Abstract Generator

© 2023 Cadence Design Systems, Inc.

Creating a User Bin in Abstract Generator

You can create user bins in addition to the existing system bins. User bins differ from system bins only in that you can delete and rename user bins, while system bins are permanent and cannot be renamed or deleted. The commands in the *Bins* menu apply only to user bins.

To create a user bin:

1. Load a library and choose *Bins – Add* from the menu bar.
2. Enter a name and set a type for the new bin by selecting a choice from the *Choose bin type* drop-down box.

   The bin type is important because it determines the default values for certain options.
3. Click *OK*.

The new bin is created at the bottom of the list in the *Bin* pane.

Related Topics

Bin Preferences for Abstract Generation

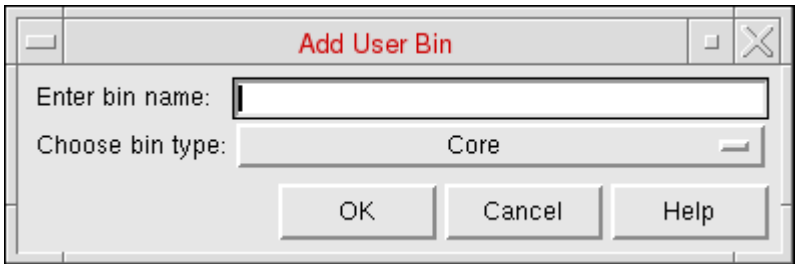
© 2023 Cadence Design Systems, Inc.

Adding a User Bin in Abstract Generator

Bins are like buckets for placing cells. Each cell in a library is always contained in one bin. Each bin is associated with certain options, which allows the cells in a library to be distributed into mutually exclusive sets for processing.

To add a new user bin to the list of existing bins:

1. Choose *Bins – Add* option.



2. Type the name of the new bin in *Enter bin name*. The bin name should not contain any spaces.
3. In *Choose bin type*, select a bin from the drop-down list. The bin type determines the default options settings for the new bin. Therefore, for example, if you create a new user bin of type *Core*, the user bin options will be identical to those currently set for the default *Core* bin.
4. Click *OK*.

Abstract Generator adds the new bin and displays it at the bottom of the list in the *Bins* pane.

Related Topics

Bin Preferences for Abstract Generation

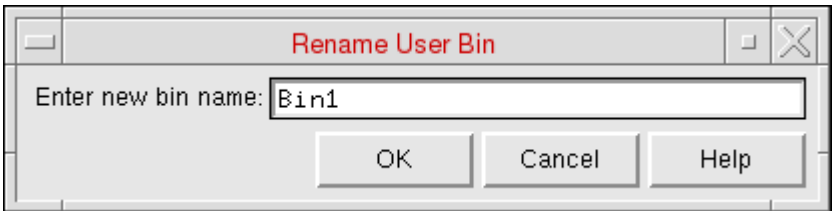
© 2023 Cadence Design Systems, Inc.

Renaming a User Bin in Abstract Generator

Bins are like buckets for placing cells. Each cell in a library is always contained in one bin. Each bin is associated with certain options, which allows the cells in a library to be distributed into mutually exclusive sets for processing.

To change the name of the selected user bin:

1. Choose *Bins – Rename*.



2. In *Enter new bin name*, type the new name for the selected user bin and
3. Click *OK* to accept the change.

Note: You cannot change the names of system bins.

Related Topics

Bin Preferences for Abstract Generation

Rename User Bin Form

© 2023 Cadence Design Systems, Inc.

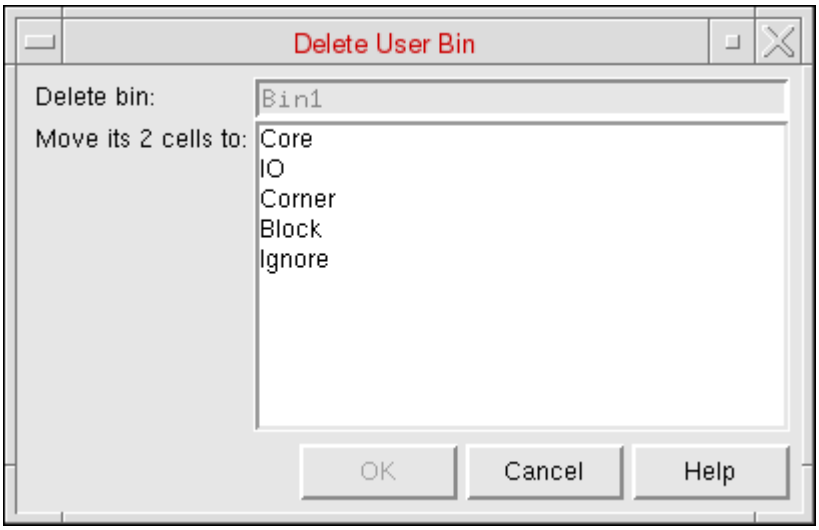
Deleting a User Bin in Abstract Generator

Bins are like buckets for placing cells. Each cell in a library is always contained in one bin. Each bin is associated with certain options, which allows the cells in a library to be distributed into mutually exclusive sets for processing

To delete a user bin:

1. Select the bin you do not need and want to delete, and choose *Bins – Delete* and confirm the deletion.

   If the bin you are deleting is not empty, you will be prompted (the Delete User Bin form will be displayed) to specify where to put its cells before the bin is deleted.



   The name of the selected user bin is displayed in the *Delete bin* field.

2. In *Move its 2 cells to*, select the bin to which you want the cells of the deleted bin to be moved
3. Click *OK*.

The cells are moved and the user bin is deleted.

Related Topics

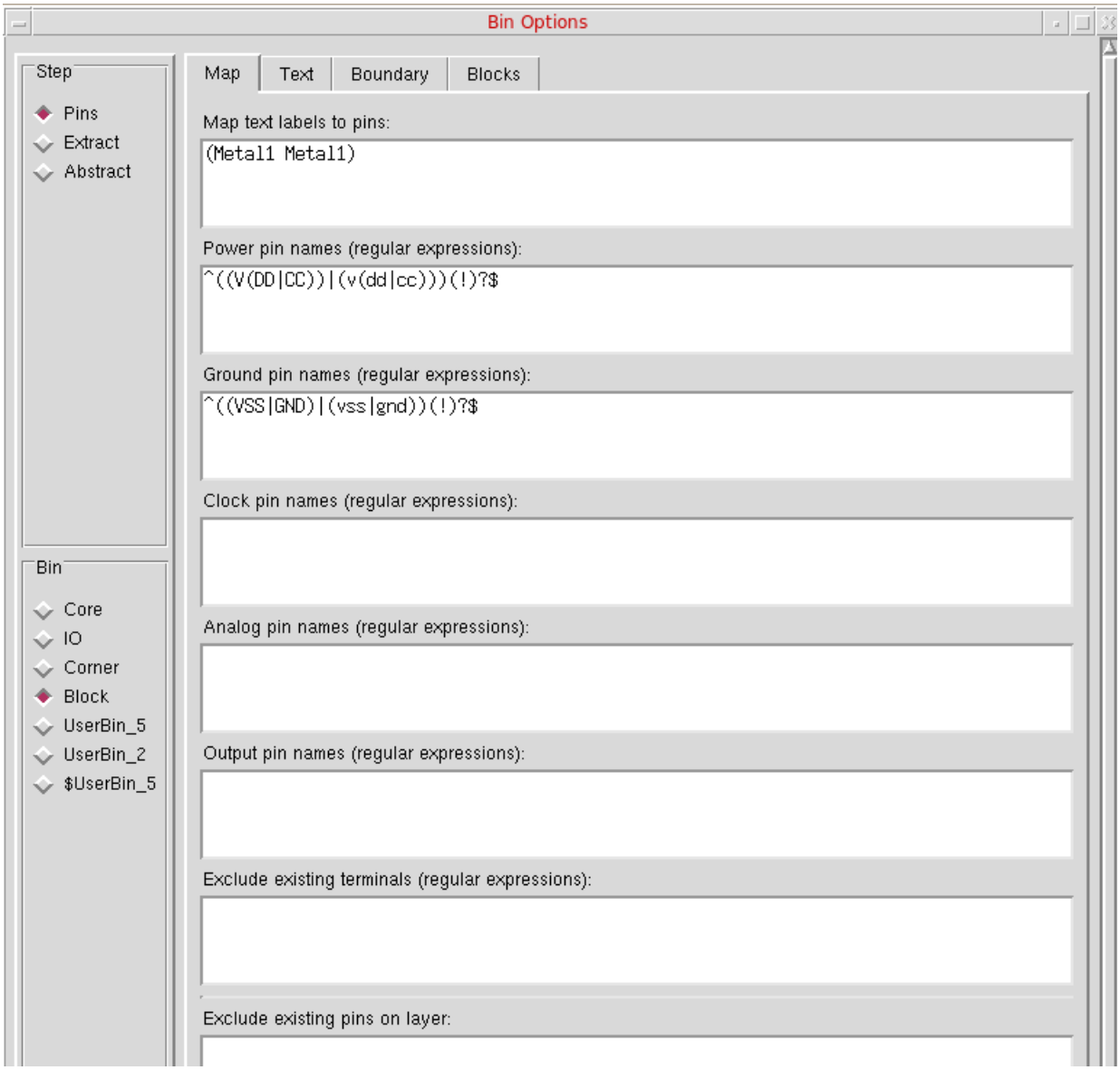
Delete User Bin Form

© 2023 Cadence Design Systems, Inc.

Setting Bin Options in Abstract Generator

The Bin Options form allows you to view and change the options associated with system and user bins. These options control the way in which Abstract Generator processes the cells contained in any given bin.

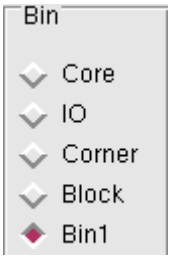
1. Choose *Bins – Options* to open the Bin Options form.



When you run flow steps on selected cells, you see a modified version of the Bin Options form, which lets you view and change options relevant to the steps about to be run.

© 2023 Cadence Design Systems, Inc.

2. In the *Bin Selector* section, use the option settings are applied to the bin specified in the panel on the left of the form.



   If there is a bin selected when you open the form, it will be selected here with the option settings for that bin displayed. If there are multiple bins selected, the first bin in the list is selected. You can view and change the options associated with any other bin by clicking the button next to the bin name.

3. In the *Step Selector* section, you can access the options associated with each step in the top left of the Bins Options form. The abstract generation flow is split into four steps: Pins, Extract, and Abstract.

   When you open the *Bin Options* form, the default selection is Pins. Use the buttons to view and modify options associated with the other steps.

4. Use the options associated with each flow step are further grouped into logical categories, accessed by clicking the appropriate tab at the top.

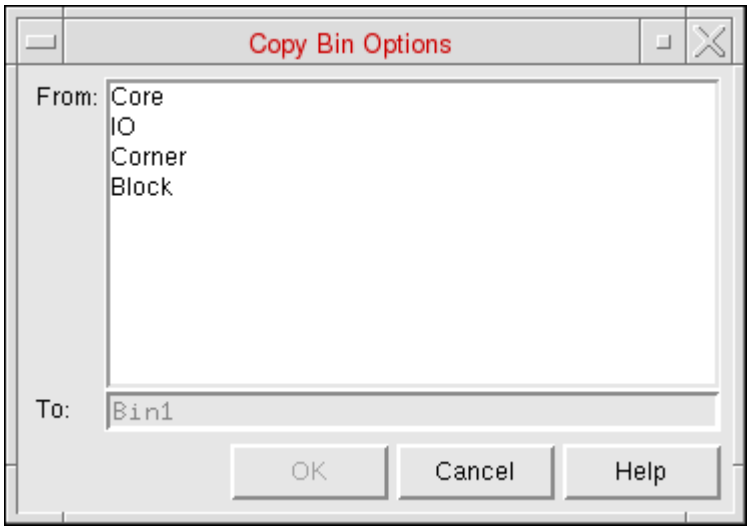
   When setting options, consider the following rules:

   - Measurements: All options that take measurements, for example for coordinates or distances, are specified in μ.
   - Layers: Where an option takes a layer name, the name supplied must be defined in the technology file; otherwise, the entry is rejected and an error is generated when you try to move from the field in question.
   - Purpose: Only purposes defined in the technology file are valid.

5. Select *Copy From Bin* to copy option settings from another bin to the selected bin. This is useful when creating a new user bin; you can selectively copy bin specific options only while copying options from one bin to another.

© 2023 Cadence Design Systems, Inc.

- Click the *Copy From Bin* button to open the Copy Bin Options form.



- In the *From* field, select the bin whose options you want to copy and click *OK*. For example, the settings for the *Core* bin could be copied to *UserBin*.
- The current bin is entered automatically in the *To* field.

This function overwrites all the current option settings for the target bin.

6. Click *OK*.

Related Topics

Copy Bin Options Form

Guidelines for Bin Options Form

© 2023 Cadence Design Systems, Inc.

Guidelines for Bin Options Form

The Bin Options form allows you to view and change the options associated with system and user bins. These options control the way in which Abstract Generator processes the cells contained in any given bin.

The following table lists the options that are disabled for specific bins and are to be excluded from that bin. This means that the options listed under a particular bin will be excluded while copying options from other bin to this bin and vice-versa.

Bin Name	Excluded Options
I/O	AbstractAdjustBoundaryPinsSig
	AbstractAdjustBoundaryPinsPwr
	AbstractAdjustRingPinsPwr
	AbstractAdjustFollowRingPin
	AbstractSiteArrayPattern
	AbstractSiteArrayCells
Core	AbstractAdjustBoundaryPinsSig
	AbstractAdjustBoundaryPinsPwr
	AbstractAdjustEdgeTowardsCore
	AbstractAdjustPowerGeometryGroupsEastWest
	AbstractAdjustClassCoreNets
	AbstractAdjustIncludeAllShapes
	AbstractAdjustCopyClassCorePort
	AbstractAdjustPinsTouchBoundary
	AbstractAdjustPinsTouchNonCoreEdge
	AbstractAdjustCreateClassBumpPort
	AbstractAdjustRingPinsPwr
	AbstractAdjustFollowRingPin

© 2023 Cadence Design Systems, Inc.

Block	AbstractAdjustEdgeTowardsCore
	AbstractAdjustPowerGeometryGroupsEastWest
	AbstractAdjustClassCoreNets
	AbstractAdjustIncludeAllShapes
	AbstractAdjustCopyClassCorePort
	AbstractAdjustPinsTouchBoundary
	AbstractAdjustPinsTouchNonCoreEdge
	AbstractAdjustCreateClassBumpPort
	AbstractSiteName
	AbstractSiteNameDefine
	AbstractSiteArrayPattern
	AbstractSiteArrayCells
	AbstractSiteArrayCellsTable
Corner	AbstractAdjustBoundaryPinsSig
	AbstractAdjustBoundaryPinsPwr
	AbstractAdjustEdgeTowardsCore
	AbstractAdjustPowerGeometryGroupsEastWest
	AbstractAdjustClassCoreNets
	AbstractAdjustIncludeAllShapes
	AbstractAdjustCopyClassCorePort
	AbstractAdjustPinsTouchBoundary
	AbstractAdjustPinsTouchNonCoreEdge
	AbstractAdjustCreateClassBumpPort
	AbstractAdjustRingPinsPwr
	AbstractAdjustFollowRingPin
	AbstractSiteArrayPattern
	AbstractSiteArrayCells
	AbstractSiteArrayCellsTable

© 2023 Cadence Design Systems, Inc.

Related Topics

Bin Preferences for Abstract Generation

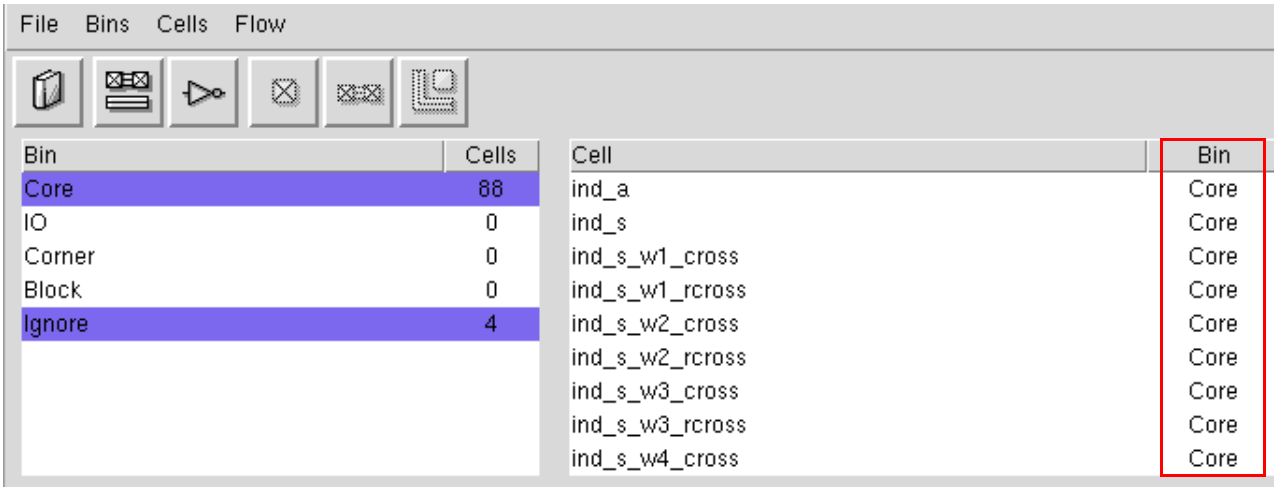
© 2023 Cadence Design Systems, Inc.

Troubleshoot System Bin Issues

ABS-11904

Failed to delete the system bin because system bins such as *Core*, *IO*, *Corner*, *Block*, and *Ignore* cannot be deleted.

Troubleshooting: The five system bins are `Core`, `IO`, `Corner`, `Block`, and `Ignore`. You cannot add, rename, or remove system bins.



ABS-11905

Failed to delete the *`name`* user bin because the bin contains cells.

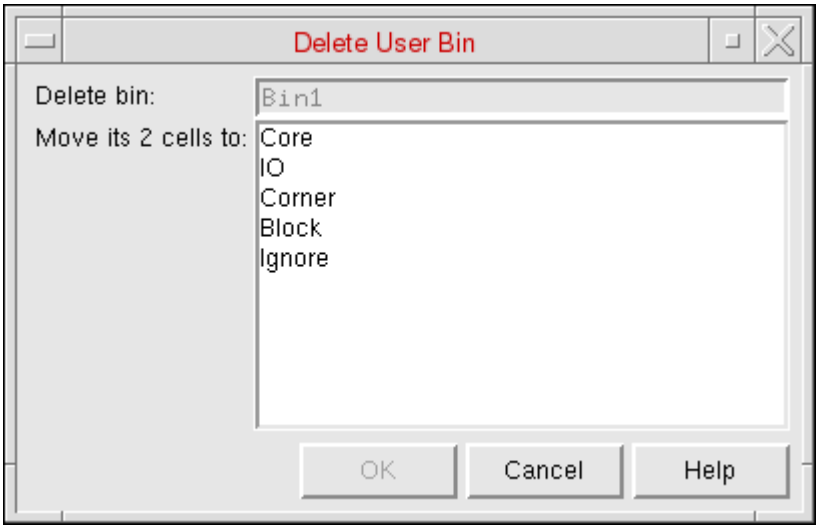
Troubleshooting:

Transfer all cells to another bin and then try again. To delete a user bin:

1. Select the bin you do not need and want to delete, and choose *Bins – Delete* and confirm the deletion.

© 2023 Cadence Design Systems, Inc.

If the bin you are deleting is not empty, you will be prompted (the Delete User Bin form will be displayed) to specify where to put its cells before the bin is deleted.



The name of the selected user bin is displayed in the *Delete bin* field.

2. In *Move its 2 cells to*, select the bin to which you want the cells of the deleted bin to be moved
3. Click *OK*.

The cells are moved and the user bin is deleted.

Related Topics

Bin Preferences for Abstract Generation

© 2023 Cadence Design Systems, Inc.

Cell Preferences for Abstract Generation

The Cell pane lists the cells contained in the selected bins and reports the status of those cells. You can use the *Cells* pane to perform sophisticated selection operations like the cell name, height, properties, and view status. When a cell library containing physical or logical information is imported into Abstract Generator, the view information about the cells is represented in the appropriate column of the *Cell* pane. Similarly, when a cell is processed through the various Abstract Generator steps – Pins, Extract, and Abstract – cellviews are created on successful completion of each step.

Abstract Generator uses the following 5.x library format cellviews:

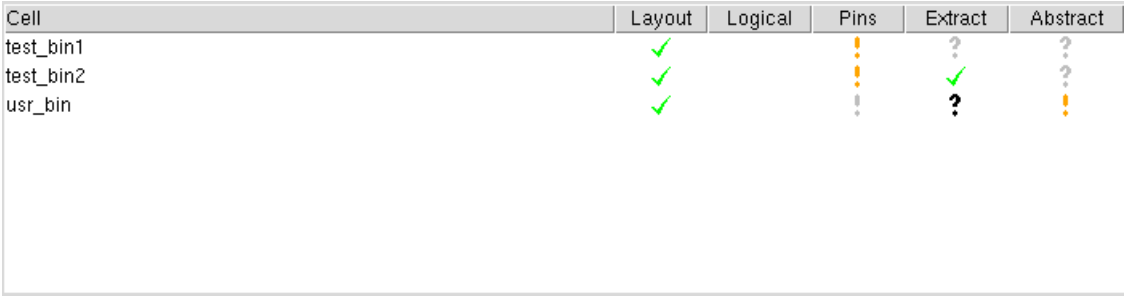
- `layout` for layout views
- `logical` for logical views
- `abstract.pin` for Pins views
- `abstract.ext` for extracted views
- `abstract` for abstract views

The *Cell* pane is divided into a number of columns, one for each of the views handled by Abstract Generator: *Layout, Logical, Pins, Extract,* and *Abstract*. A symbol in any of these columns indicates the presence of that view for the selected cell.

Note: An additional column, *Bin*, appears as the first column when more than one bin is selected in the *Bin* pane. The *Bin* column displays the name of the bin to which the cell belongs.

© 2023 Cadence Design Systems, Inc.

The symbol displayed indicates the status of that view, as summarized in the following table.



Symbol	What It Represents
✓	A green tick indicates that the view is valid.
!	An amber exclamation mark indicates that the view was generated with warnings.
✗	A red cross indicates that the view was generated with errors.
?	A black question mark indicates that the view is present but was not created by Abstract Generator.

There is also a grayed-out version of each symbol, which indicates that the cellview in question has been rendered invalid, typically because of a change to one of the option settings or because of the regeneration of a predecessor view when an earlier step in the flow is re-run.

For example, a gray cross against a cell means that the view in question was originally generated with errors but has now been rendered invalid.

Note: To maximize the usefulness of the feedback, cell status is not propagated from one view to another. For example, the pins view for a particular cell can show warnings against it, while the extract view for the same cell has none.

The *Cells* menu contains commands to perform the following tasks:

- Selecting Cells Based on a Criteria in Abstract Generator
- Selecting All Cells
- Deselecting All Cells
- Revalidation of Cells with Invalid Views in Abstract Generator
- Sorting Cells in Abstract Generator

© 2023 Cadence Design Systems, Inc.

- Distributing Cells into Bins in Abstract Generator
- Moving Cells Across Bins in Abstract Generator
- Viewing and Editing Cell Properties
- Viewing and Editing Cell Terminal Properties in Abstract Generator
- Viewing a Cell Status Report in Abstract Generator
- Editing Cellviews in Virtuoso Layout Editor in Abstract Generator

Related Topics

Standalone Abstract Generator

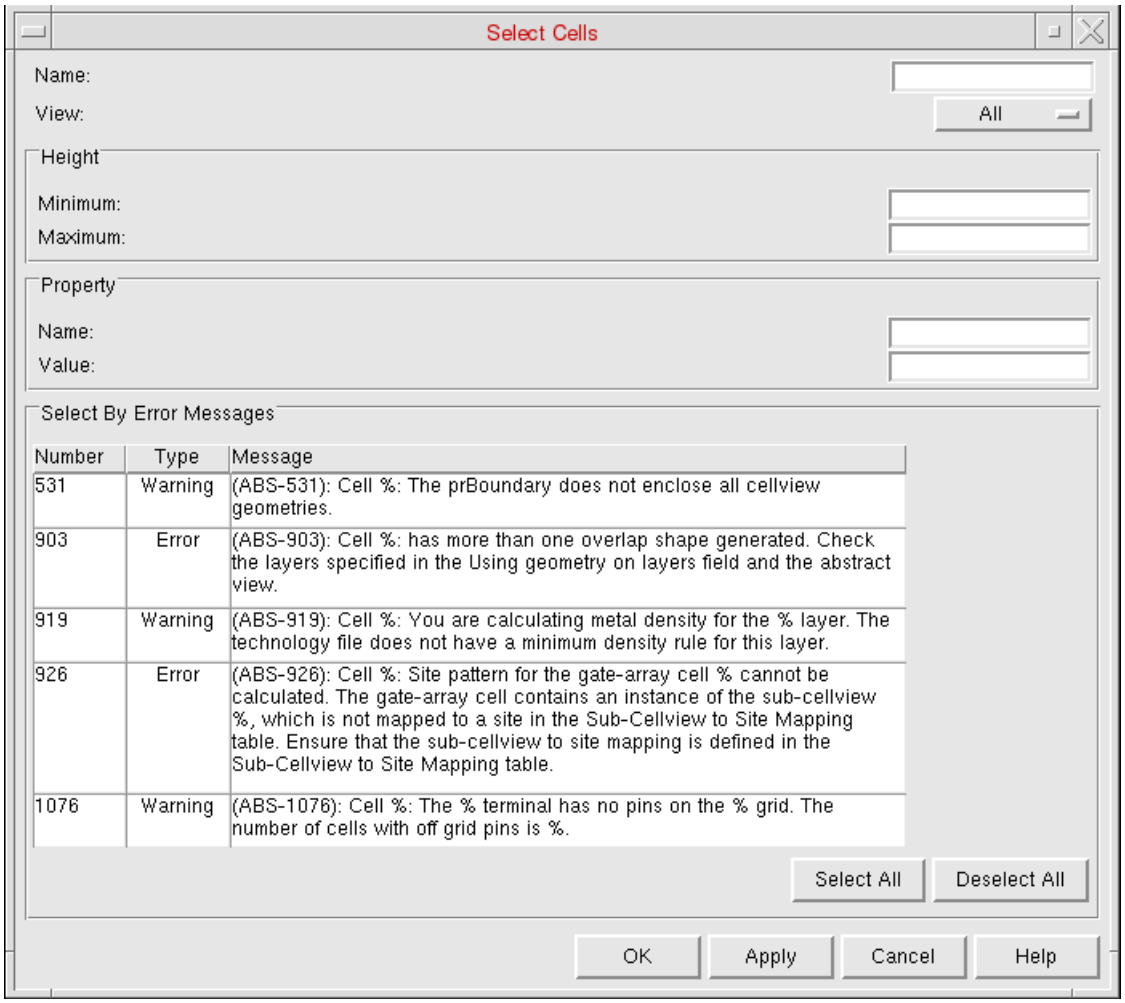
© 2023 Cadence Design Systems, Inc.

Selecting Cells Based on a Criteria in Abstract Generator

The Select Cells form allows you to selects one or more cells using a variety of selection criteria.

1. Choose *Cells – Select* to start this command.

   This displays the Select Cells form.



2. Select *Name* to enter a regular expression that is included in the cell names you want to select. All matching cells are selected, irrespective of the *View* set.

3. Select *View* to choose the view you are interested in. Only cells that have the specified *View* present and match any subsequent *Height*, *Property* or *Status* criteria are selected.

© 2023 Cadence Design Systems, Inc.

4. Select *Height* to specify minimum or maximum cell heights, or a range within which the cell height must fall.
5. In *Property,* enter a system or user-defined cell property and a value for the cells you want to select.
6. Use *Select By Error Messages* to select one or more of the error messages listed in the table to select cells in a library. Only cells that have had a specific, selected error message applied to them will be filtered through in the selection procedure.

   The figure shown above displays a list of error messages based on which you can select cells.
7. Click *OK* to select the cells matching the criteria and close the form, or use the *Apply* button to perform incremental selection without closing the form.

Note: The criteria specified are applied only to the cells in the current bin.

Selecting All Cells

The *Cells – Select All* command selects all cells displayed in the *Cell* pane. Clicking the table header of the *Cell* pane also selects all the cells.

This function does not select cells in bins that are not selected.

Deselecting All Cells

When you click *Cells – Deselect All*, no cell in the cells pane is selected.

Related Topics

Select Cells Form

© 2023 Cadence Design Systems, Inc.

Revalidation of Cells with Invalid Views in Abstract Generator

The *Cells – Revalidate Selected* command revalidates all invalid views for the cells selected in the *Cell* pane.

Abstract Generator invalidates a view if the view is out of date. It then turns the symbol gray for that view in the *Cell* pane. Abstract Generator does not allow you to proceed with the abstraction flow until you run the step again to create that view.

A view might become invalid in any of the following situations:

- The technology file is changed after the view was created.
- The preceding view in the flow has a newer timestamp.
- Abstract options for the step have changed after the view was created.
- The cell is moved to another bin.

In these situations, Abstract Generator, by default, invalidates the view to enforce consistency between the techfile, changed cellviews, and options. This helps the inexperienced user. However, in some cases this is overly restrictive for experienced users. The *Revalidate Selected* option provides the override functionality to these experienced users.

Note: The result of selecting the *Revalidate Selected* command is that the view status is no longer grayed out even if the view is invalid. However, the errors associated with the view would still need to resolved prior to running any subsequent steps.

The need for using the *Revalidate Selected* option arises from the situation in which you might want to process the cells in a single bin with slightly different options. To do this without the *Revalidate Selected* option, you would need to create a user bin for each cell, move cells to the different user bins and then process the abstract for each bin. With the *Revalidate Selected* option, you can retain the cells in a single bin, modify the options for them, and process the abstract as required before revalidating views.

Related Topics

Cell Preferences for Abstract Generation

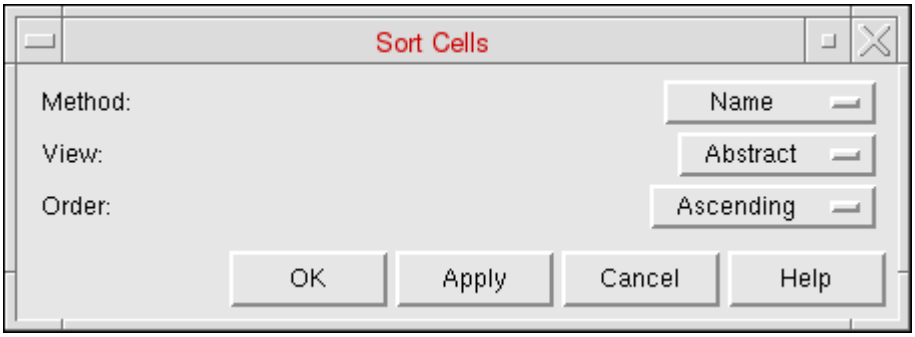
Selecting Cells Based on a Criteria in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Sorting Cells in Abstract Generator

The Sort Cells form sorts the selected cells according to a variety of sorting criteria. The specified criteria are applied only to the selected cells displayed in the *Cell* pane and not to the overall library.

1. Choose *Cells – Sort* command to open the Sort Cells form.



2. In *Method*, select criterion for sorting from *None, Name, Height, Status,* or *Selection*.
3. In *View*, select from *Layout*, *Logical*, *Pins*, *Extract*, or *Abstract*.

   The *Height* and *Status* methods consider only the view specified here. Cells that do not have the specified view are not sorted.
4. In *Order*, select *Ascending* or *Descending*.
5. Click *OK* to sort the cells when you have specified your criteria.

Related Topics

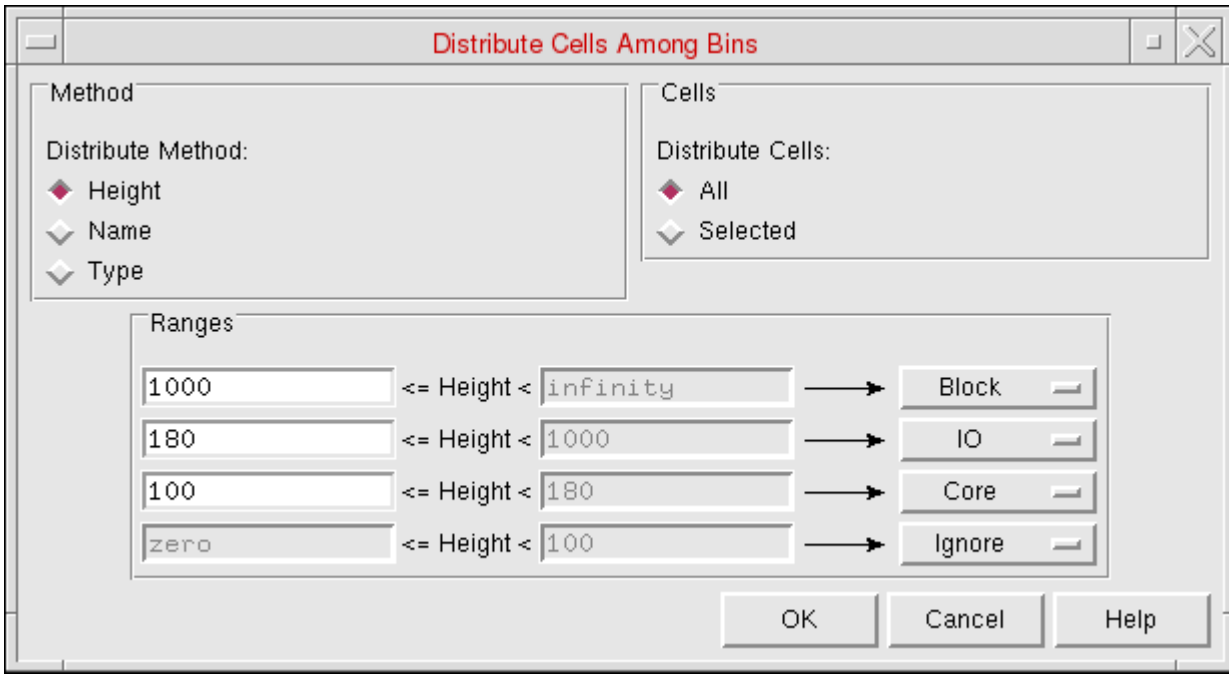
Sort Cells Form

© 2023 Cadence Design Systems, Inc.

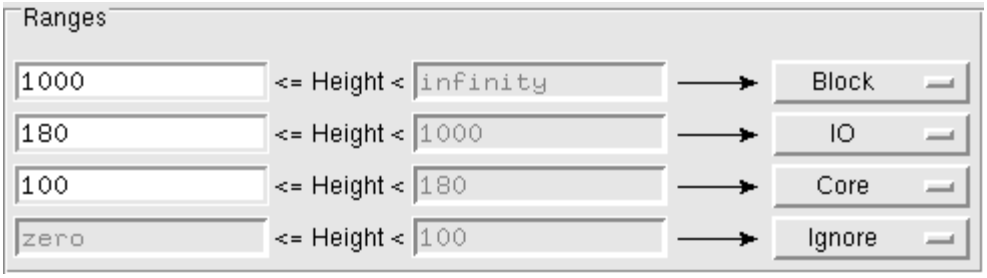
Distributing Cells into Bins in Abstract Generator

Use the Distribute Cells Among Bins form to distribute cells among the system and user bins according to cell height, name, or type.

1. Choose *Cells – Distribute* to open the Distribute Cells Among Bins form.



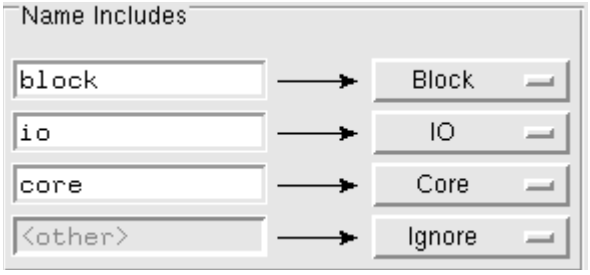
2. The *Method* group options allow you to specify the criteria you want to use. Choose from *Height*, *Name* and *Type*.
3. Selecting the *Height* method changes the lower part of the form to display the following:



You can specify a series of height ranges in μ, and the bins to which cells falling within a particular range are to be moved. To ensure that no cells are missed, only the lower bound of each range is editable. When you change the lower bound of one range, the upper bound of the next range down is adjusted automatically.

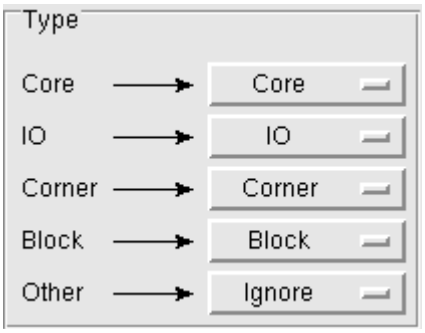
© 2023 Cadence Design Systems, Inc.

4. Selecting the *Name* method changes the lower part of the form to display the following:



You can specify strings of characters and target bins for cell names containing those strings. For example, the settings shown above would move all the cells with names that include the string *block* to the *Block* bin, those including the string *io* to the *IO* bin, and so on. Any cells not covered by specific criteria fall into the “other” category and are moved to the bin specified.

5. Selecting the *Type* method changes the lower part of the form to display the following:



Using this form, you can distribute the cells to other system and user bins. The *Type* method distribution is based on the `CellType` attribute of a layout cell on OpenAccess.

The first column represents the `CellType` attribute. By default, the `CellType` attributes in the first column are mapped to the corresponding system bins in the second column. The choices in the second column corresponding to a specific type are restricted to the user-defined and system bins of the same type. For example, if *myBin* is a user-defined bin of the type *Block*, *myBin* would be available for selection with the *Block* system bin in the second column. For cellviews with an unknown `CellType` (*Other*), all system and user-defined bins come up as options in the *Ignore* cyclic list.

6. In the *Cells* group, select whether you want to distribute *All* the cells currently displayed in the *Cell* pane or limit the operation to the set of cells *Selected* when you opened the form.

7. Click *OK* to distribute the cells when you are satisfied with the height ranges.

© 2023 Cadence Design Systems, Inc.

Related Topics

Distribute Cells Among Bins Form

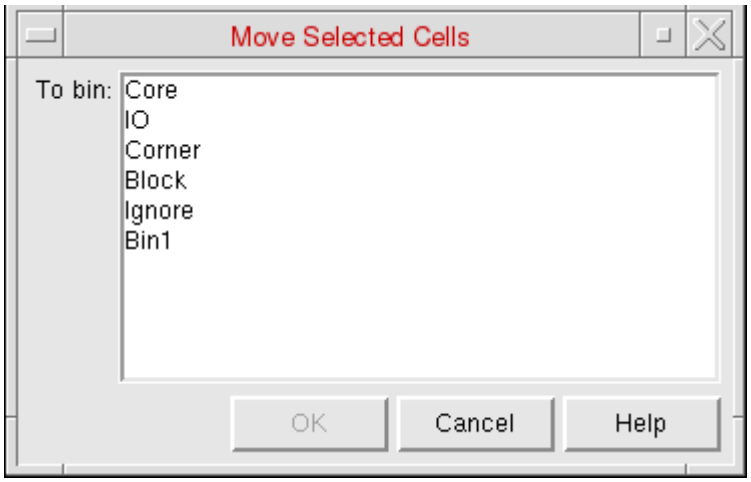
© 2023 Cadence Design Systems, Inc.

Moving Cells Across Bins in Abstract Generator

The Move Selected Cells form moves one or more selected cells to a different bin.

1. In the *Cell* pane, select the cells you want to move either manually or by using *Cells – Select,* then choose *Cells – Move*.

   This displays the Move Selected Cells form.



2. Choose the target bin and click *OK*.

The selected cells are moved into the selected bin. The status for all the views remains unchanged.

Related Topics

Move Selected Cells Form

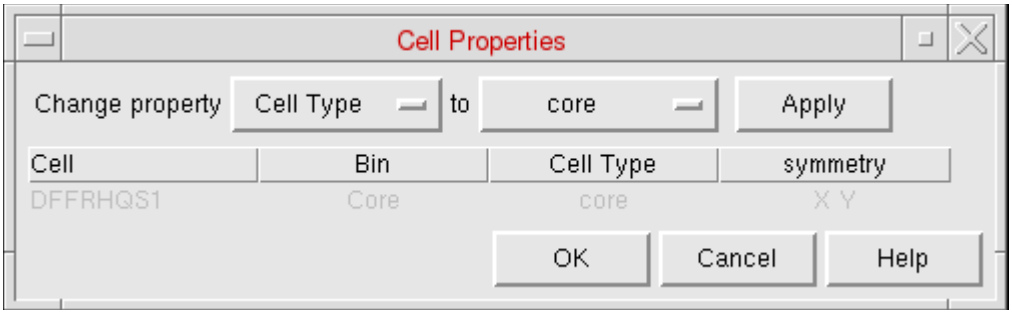
© 2023 Cadence Design Systems, Inc.

Viewing and Editing Cell Properties

You can use the Cell Properties form to view and change the properties on cells.

To change a property of a cell:

1. Select *Cells – Cell Properties* after selecting one or more cells in the *Cell* pane.



   The selected cells are listed in the Cell Properties form. Also listed is the bin in which the cell is stored and the current settings for the *prCellClass* and *symmetry* properties. You can select more than one cell by using the usual `Shift` and `Ctrl` key bindings.

Important

   Abstract Generator is able to set properties only on valid views that it has created (Pins, Extract and Abstract). If a cell contains none of these views or if the views are invalid, the entries in the table are grayed out and you cannot change the properties for that cell.

2. Select one or more cells in the list and use the *Change property* cyclic field to select the property you want to change.

   This changes the *Allowed Values* in the second cyclic field.

3. Select the new value for the selected cells.
4. Click *Apply* to update the form. This will not update the database.
5. Click *OK* to accept all your changes and close the form.

Related Topics

Cell Properties Form

© 2023 Cadence Design Systems, Inc.

Valid Values for the prCellClass and prCellClass Properties in Abstract Generator

prCellClass

The allowed values for the *prCellClass* property depend on the bin in which the cell resides. The table below shows the allowed *prCellClass* entries and the corresponding LEF descriptions for each bin.

Bin	Value	LEF
Core	`core`	`CLASS CORE`
	`feedthru`	`CLASS CORE FEEDTHRU`
	`tieHigh`	`CLASS CORE TIEHIGH`
	`tieLow`	`CLASS CORE TIELOW`
	`spacer`	`CLASS CORE SPACER`
	`antennaCell`	`CLASS CORE ANTENNACELL`
	`preEndcap`	`CLASS ENDCAP PRE`
	`postEndcap`	`CLASS ENDCAP POST`
	`wellTap`	`CLASS CORE WELLTAP`
IO	`pad`	`CLASS PAD`
	`input`	`CLASS PAD INPUT`
	`output`	`CLASS PAD OUTPUT`
	`inOut`	`CLASS PAD INOUT`
	`power`	`CLASS PAD POWER`
	`spacer`	`CLASS PAD SPACER`
	`areaIO`	`CLASS PAD AREAIO`

© 2023 Cadence Design Systems, Inc.

Bin	Value	LEF
Block	`block`	`CLASS BLOCK`
	`cover`	`CLASS COVER`
	`ring`	`CLASS RING`
	`coverBump`	`CLASS COVER BUMP`
	`blackBox`	`CLASS BLACKBOX`
	`soft`	`CLASS BLOCK SOFT`
Corner	`bottomLeftEndcap`	`CLASS ENDCAP BOTTOMLEFT`
	`bottomRightEndcap`	`CLASS ENDCAP BOTTOMRIGHT`
	`topLeftEndcap`	`CLASS ENDCAP TOPLEFT`
	`topRightEndcap`	`CLASS ENDCAP TOPRIGHT`

symmetry

Specifies the symmetry for the selected cells.

R0 | *X* | *Y* | *X Y*

Note: The symmetry set using this option overrides all other symmetry settings on the cell.

Related Topics

Cell Properties Form

© 2023 Cadence Design Systems, Inc.

Viewing and Editing Cell Terminal Properties in Abstract Generator

The Terminal Properties form lets you edit terminal properties for the cells in your library. Abstract Generator assigns the properties to the terminals in the database. If the logical information already contains these properties on terminals, Abstract Generator displays them in the Terminal Properties form and propagates them through various Abstract Generator views. This form works in the same way as the Cell Properties Form.

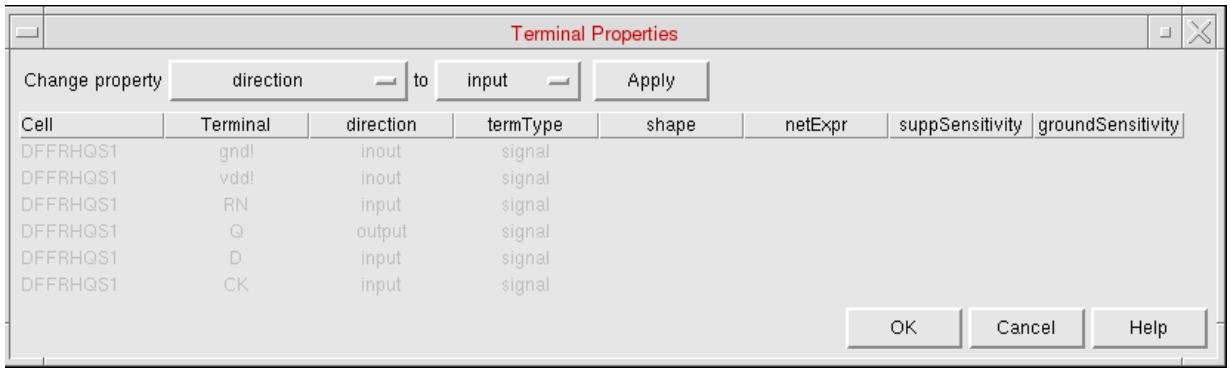
Important

Abstract Generator is able to set properties only on valid views that it has created (Pins, Extract and Abstract). If a cell contains none of these views or if the views are invalid, the entries in the table are grayed out and you cannot change the properties for the terminals in question.

To change a terminal property of a cell, perform the following steps:

1. Choose *Cells – Terminal Properties* after selecting one or more cells in the *Cell* pane.

   The Terminal Properties form is displayed. The table contains an entry for each terminal (*Terminal* column) defined for each of the selected cells (*Cell* column) selected in the *Cell* pane.



2. Select one or more terminals from the list and use *Change property* to select the property you want to change.This changes the Allowed Values in the second cyclic field.
3. Select the new value for the selected cells.
4. Click *Apply* to make the change. This updates the form but not the database.
5. Click *OK* to accept all your changes and close the form.

Note: Terminal property settings are saved in the view of the current step. Therefore, when a step is run again, the terminal properties will be restored to the original values set prior to any user amendments.

© 2023 Cadence Design Systems, Inc.

Allowed Values

Field	Allowed Values
direction	`input`
	`output`
	`inout`
	`jumper`
	`tristate`
	`unused`
termType	`signal`
	`clock`
	`analog`
	`power`
	`ground`
shape	`feedthru`
	`abutment`
	`ring`
netExpr	Specify a valid terminal net expression in the following format:
	`[@gnd:%:gnd!]`
	To remove a net expression, select the terminal, click the *netExpr* column header and clear the field at the top of the form.
suppSensitivity	Power terminal names.
groundSensitivity	Ground terminal names. See .

Related Topics

Terminal Properties form

Support for Inherited Connections in Abstract Generator

Understanding Connectivity and Naming Conventions.

© 2023 Cadence Design Systems, Inc.

Support for Inherited Connections in Abstract Generator

Abstract Generator provides support to inherited connections for managing multiple power and voltage supplies in designs through the properties *netExpr*, *suppSensitivity*, and *groundSensitivity*.

To set the supply sensitivity for a signal pin, the cell that contains the signal pin must have more than one power terminal. Similarly, to set the ground sensitivity for a signal pin, the cell containing the signal pin must have more than one ground terminal.

The value of the *suppSensitivity* property is displayed in the view of the current step only when you set the value after running the step. For example, if you want to see the property in Pins view, you must set its value after running the Pins step. To get information about the supply sensitivity in a LEF file, you must set its value after running the Abstract step and then export the LEF file.

The Terminal Properties form displays all power and ground terminals of the cells selected in the *Cell* pane. You can select single or multiple signal pins on the same or different cells to set their supply or ground sensitivities. The property value will be updated on a signal pin only if all of the following criteria hold true:

- The cell with the signal pin has more than one power (for setting supply sensitivity) or ground (for setting ground sensitivity) terminal.
- The value to set is a power (for setting supply sensitivity) or ground (for setting ground sensitivity) terminal on the cell with the signal pin.

If either of these conditions is not met, Abstract Generator will issue an appropriate warning message when you click *Apply*.

Consider a situation in which `CellA` and `CellB` have the following power, ground, and signal terminals:

Cell	Terminal	termType
`CellA`	`VDD`	`power`
	`VSS`	`ground`
	`in1`	`signal`
	`in2`	`signal`

© 2023 Cadence Design Systems, Inc.

Cell	Terminal	termType
CellB	VDD	power
	VSS	ground
	out1	signal
	out2	signal
	PWR	power
	GND	ground

In this situation, Abstract Generator populates the values VDD and PWR for the *suppSensitivity* property and the values VSS and GND for the *groundSensitivity* property.

Notice that CellA contains only one power and one ground terminal. So, it is not feasible to update the supply or ground sensitivity for the signal pins on this cell. CellB contains more than one power and ground terminals. To set the supply sensitivity on a signal pin of this cell, you can choose one of VDD or PWR. Similarly, to set the ground sensitivity on a signal pin of this cell, you can choose one of VSS or GND.

The following table summarizes the values that can be set for the *suppSensitivity* and *groundSensitivity* properties for the signal pins on CellA and CellB.

Cell	Terminal	termType	suppSensitivity	groundSensitivity
CellA	VDD	power		
	VSS	ground		
	in1	signal	X	X
	in2	signal	X	X
CellB	VDD	power		
	VSS	ground		
	out1	signal	VDD, PWR	VDD, GND
	out2	signal	VDD, PWR	VDD, GND
	PWR	power		
	GND	ground		

© 2023 Cadence Design Systems, Inc.

'X' indicates that values cannot be set for *suppSensitivity* and *groundSensitivity* properties.

Related Topics

Viewing and Editing Cell Terminal Properties in Abstract Generator

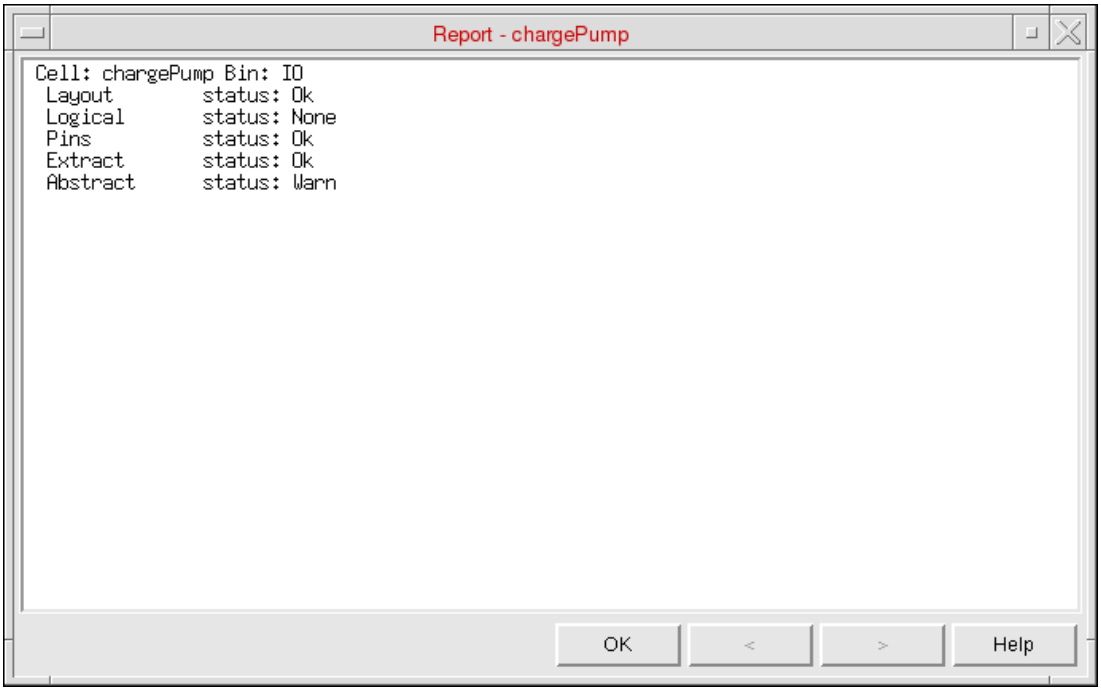
© 2023 Cadence Design Systems, Inc.

Viewing a Cell Status Report in Abstract Generator

The cell report provides detailed information about the status of a cell, with explanations of any messages generated for the different views present. Only views that have a valid status are reported.

To view a cell report:

1. Select the cell or cells for which you wish to view the report.
2. Choose *Cells – Report* from the menu.



3. If you select more than one cell, the arrow buttons at the bottom of the form allow you to move backward and forward through the selected set.

In the above figure, the arrow buttons are not enabled because only one cell has been selected in the *Cell* pane.

Related Topics

Editing Cellviews in Virtuoso Layout Editor in Abstract Generator

© 2023 Cadence Design Systems, Inc.

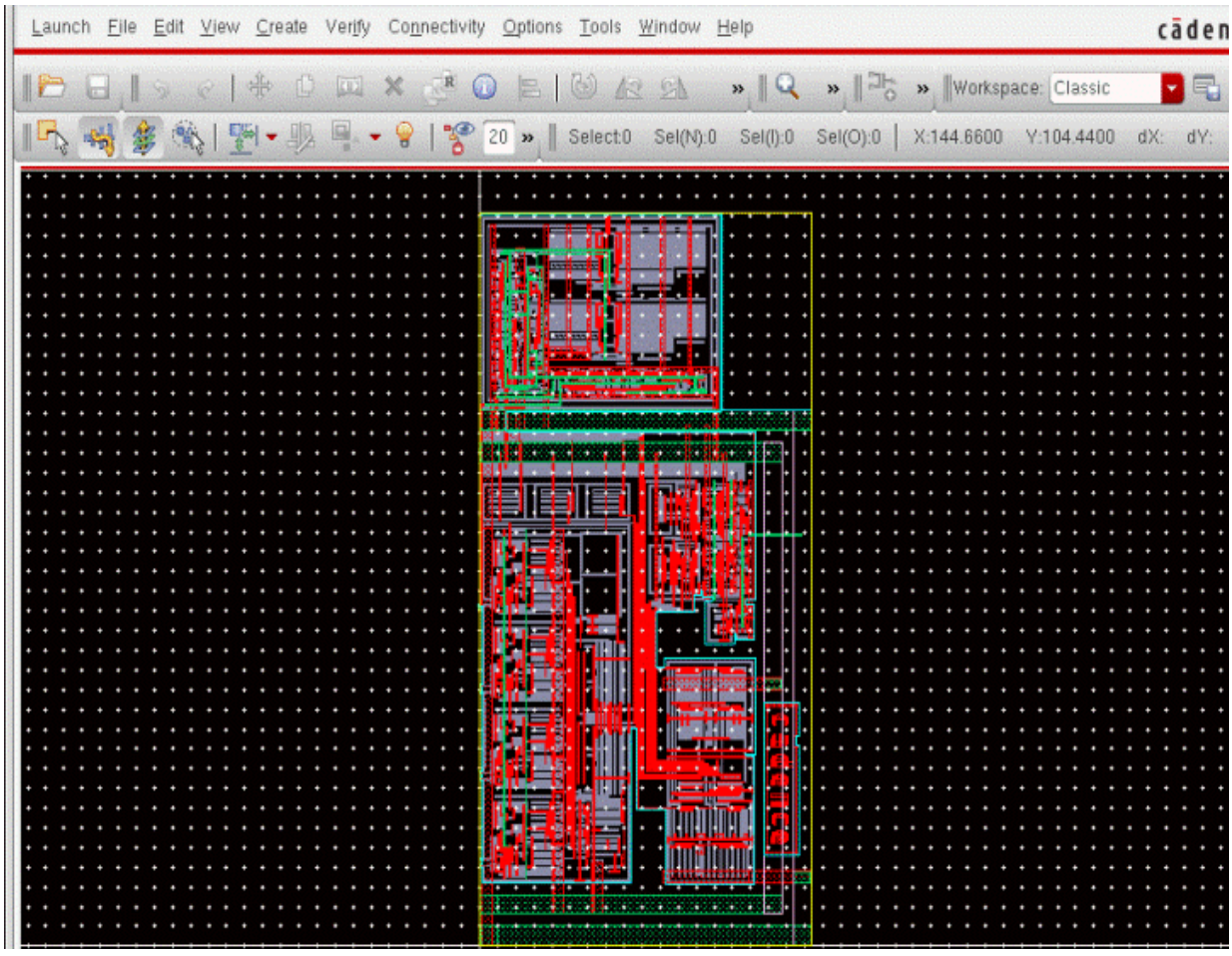
Editing Cellviews in Virtuoso Layout Editor in Abstract Generator

You can edit cellviews in Virtuoso® Layout Suite XL in Abstract Generator. Before editing a cellview, ensure that you have the correct Virtuoso Layout Suite XL license and it is set in your path.

To view and edit any cellview that exists for a selected cell:

➡ Choose *Cells – Edit – Layout*.

When you do this, Abstract Generator invokes Virtuoso Layout Suite XL Editor.



You can choose to view and edit the following cellviews: *Layout*, *Pins*, *Extract*, and *Abstract*. You cannot edit a selected cell that does not have an available view or the view is invalid. Moreover, if a view is set to be *read-only*, the Layout Editor will also only be available in read-only mode for that view.

© 2023 Cadence Design Systems, Inc.

Related Topics

Layout Editor

© 2023 Cadence Design Systems, Inc.

Troubleshoot Cell Issues

This section lists the common problems found in cells in the Virtuoso Studio design environment and provides information about how to fix these to enable Abstract Generator to run.

ABS-11001

Cell *name*: The *view name* view has been removed.

Cellviews are created upon the successful completion of each flow step in the abstract generation process. This message is generated when a cell view has been removed outside of Abstract Generator.

Troubleshooting: You can recreate the missing views by importing either GDSII (*File – Import – Stream (GDSII)*) or DEF data (*File – Import – DEF*). Alternatively, you can run the appropriate abstract generation steps.

ABS-11004

Cell *name*: Failed to open the *view name* view because *message*.

This error is issued when you try to open a non valid cell view.

Troubleshooting: Check the cell and remove from it the library if required.

ABS-11011

Cell *name:* The direction of the *name* terminal has changed from *direction* to *direction*. This change will invalidate any abstract generation steps already run. The terminal directions can be modified in the Terminal Properties form.

ABS-11012

Cell *name:* The type of the *name* terminal has changed from *termType* to *termType*. This change will invalidate any abstract generation steps already run. The terminal type can be modified in the *Terminal Properties* form.

Troubleshooting: If this change was not intentional, use the Terminal Properties form to restore the terminal type to its original setting.

© 2023 Cadence Design Systems, Inc.

ABS-11013

Cell *name*: The shape of the *name* terminal has changed from *shape* to *shape*. This change will invalidate any abstract generation steps already run. The terminal shape can be modified in the *Terminal Properties* form.

Troubleshooting: If this change was not intentional, use the Terminal Properties form to restore the terminal type to its original setting.

ABS-11024

Failed to process the cell because the *step name* view *reason*. Check the cell directory and close all applications that is using the same step and then try again.

Troubleshooting: Check the cell directory and remove any locked files or close an application that is using the same flow step.

Related Topics

Viewing and Editing Cell Terminal Properties in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Abstract Generation in Batch Mode

During an abstract session, every command and menu option you use is recorded in the `abstract.record` file. You can use this file as the replay file for the next session in either GUI or non-GUI mode. The replay file can be used to rerun the previous session or run a new session in batch mode. For this, you only need to change the library and cell names in the replay file.

Note: To save your previous work and to avoid the `abstract.record` file from being overwritten, the `abstract.record` file is renamed as `abstract.replay`, and then it is replayed. A new `abstract.record` file is created for the current session.

To run Abstract Generator in batch mode:

- For GUI mode, run the following command at the command line prompt.

  ```
  % abstract -replay filename
  ```

- For non-GUI mode, run the following command at the command line prompt.

  ```
  % abstract -nogui -replay filename
  ```

Abstract Generator runs the steps in sequence as per the replay file settings. After the batch run is complete, a library is created in the Library Manager, and the logical, layout, and abstract views of the selected cells are imported into the library. An LEF file is also generated depending on the selection of the GUI or non-GUI mode.

Note:

- It is recommended to remove any `.abstract` files between the GUI and non-GUI replay runs.
- In the non-GUI mode, it may be necessary to explicitly set the default bin in the replay file.

  ```
  absSetOption("DefaultBin" "Core")
  ```

This command must be inserted before the `absSelectCell()` call so that the selected cells can be imported into the default bin.

- In the `abstract.record` (replay) file, you can define the same cell and terminal properties for multiple cells belonging to a single bin or different bins. For this, you only need to add a `foreach` statement with the selected cells. For example,

  ```
  absExtract()
  celllist_for_prop_add = cellList("cell1" "cell2")
  cellList=absGetSelectedCells()
  cellList=parseString(cadr(parseString(cellList ":")))
  ```


© 2023 Cadence Design Systems, Inc.

```
                foreach(cell cellList
                          absSetCellProp(cell "symmetry" "X Y R90")
                          absSetCellProp(cell "prCellClass" "areaIO")
                          absSetTerminalProp( cell "VDD" "shape" "abutment")
                          absSetTerminalProp( cell "VSS" "shape" "abutment")
                         );foreach
        absAbstract()
        absExportLEF()
```

Here, you must run the Pin and Extract steps before selecting the cells for defining the cell and terminal properties.

- While using the replay file, Abstract Generator generates LEF abstracts for cells of both the previous and current sessions. If you want LEF abstracts for only the current session, do one of the following:
  - Move the cells of the previous session to the *Ignore* bin.
  - Move the cells of the current session to a different bin, generate LEF abstracts only for the cells of that bin, and then place the cells back in the original bin.
- In an abstract session, the bin-specific and step-specific options are saved in the `.abstract.options` file of a library. If you do not want these settings in batch mode, update the `abstract.record` (replay) file so that the library defaults are skipped. The `.abstract.options` file does not exist if you have not run Abstract Generator on the library.

Related Topics

Running Virtuoso Abstract Generator Using Replay File

Abstract Generator Log, Replay, and Record Files

Recording Commands and Events in Abstract Generator

Replaying Commands in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Abstract Generator Log, Replay, and Record Files

During an abstract session, the `abstract.log`, `abstract.record`, and `abstract.replay` files are created. In all cases, these files will have the same numbering to identify the correct abstract shell.

File Name	Description
`abstract.log file`	Stores all the information commands and the info, warning, and error messages issued by Abstract Generator during the current session. Instead of using the default abstract.log file, you can use the -log option to specify a different log file name.
Example	To create a log file named session.log, specify the following: `% abstract -log session.log` If there are multiple active abstract sessions in the same test case directory, then separate log files with names such as abstract.log, abstract.log.1, and abstract.log.2 are created.
`abstract.log.cdslck file`	Along with a log file, an abstract.log.cdslck file is also created for each new abstract session. An abstract.log.cdslck file indicates that an abstract session is currently running, for which the log file is being used. The abstract.log.cdslck file exists only for the current abstract session, and is deleted when you exit the session. The abstract.log is overwritten if there is no abstract.log.cdslck file present in the current directory. If you specify a different log name for a session, then the corresponding <user_defined>.log.cdslck is created.
Example	For session1.log file, a corresponding session1.log.cdslck file is created. If a session1.log.cdslck already exists, and you invokes an abstract session by specifying: `% abstract –log session1.log` In such cases, an error is displayed and Abstract Generator is not invoked. To invoke Abstract Generator, you either need to specify a different log file name or close the currently active session and try again.
`abstract.record`	Every command and option you use during the current abstract session is recorded in a .record file. The numbering of the abstract.record and abstract.log files is always in sync.

© 2023 Cadence Design Systems, Inc.

`abstract.replay` For every abstract session, an abstract.record file is created. You can use the abstract.record file from a previous run as the replay file for the current session by specifying:

```
% abstract -replay abstract.record
```

To save your previous work and to avoid the abstract.record file from being overwritten, the abstract.record file is renamed as abstract.replay, and then it is replayed. A new abstract.record file is created for the current session.

Related Topics

Conversion of Common Lisp Files to SKILL or TCL Format

Options and Status Files in Abstract Generator

Specifying the CDS.log File Name

Message Filters in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Specifying the CDS.log File Name

To specify a unique log file name using the `CDS_LOG_VERSION` environment variable:

➡ Set the variable before starting the Cadence software or include the `setenv` command in your `.cshrc` file:

```
setenv CDS_LOG_VERSION {sequential | pid}
```

Here:

- `sequential` adds a sequential number to the name of the log file, such as CDS.log.1 or CDS.log.2
- `pid` adds the number of the UNIX process to the name of the log file, such as CDS.log.1719 or CDS.log.2250

Related Topics

Conversion of Common Lisp Files to SKILL or TCL Format

Options and Status Files in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Message Filters in Abstract Generator

You can use the following option to filter redundant messages. Only info and warning messages are suppressed with this option.

```
absSetOption(suppressMessage <id1> <id2> <id3> …)
```

Abstract Generator suppresses the messages with the message IDs mentioned in `suppressMessage`. The suppressed messages are not displayed in the Abstract Log window and not stored in the `abstract.log` file. An info message is displayed for the first time for every suppressed message. At the end of the abstract step, a table displays the messages that have been suppressed along with their suppression times during the current session.

Note: Error messages are not suppressed with the above option. If an error message ID is mentioned in `suppressMessage`, then a warning message displays stating that this is an error message and will not be suppressed.

Related Topics

Conversion of Common Lisp Files to SKILL or TCL Format

Options and Status Files in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Exporting a Report in Abstract Generator

The report function outputs a summary of the cells in your library. It contains statistical information about the whole library, such as the number of cells by type and the number of cells with errors or warnings, and more detailed information about each of the individual cells.

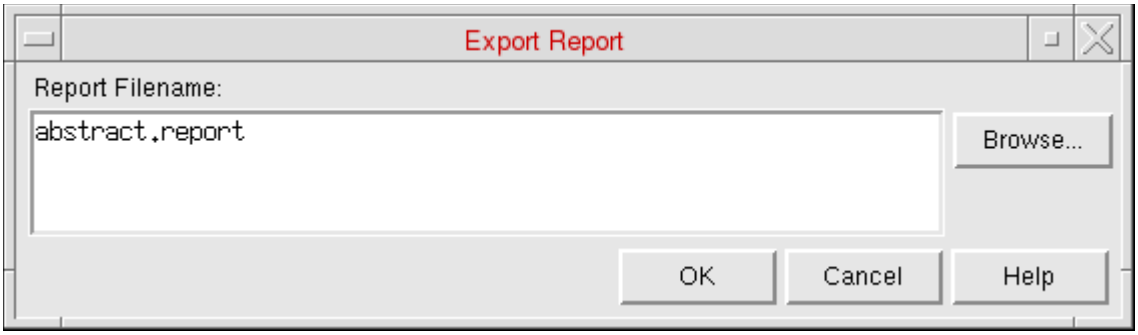
The report file contains:

- General information about the tool version
- Date and time the report was generated
- Details of the number of errors and warnings generated during the current session
- Total number of cells by type
- Number of cells with errors or warnings
- Number of cells with valid logical views but no layout views

To export a report:

1. Choose *File – Export – Report* to start this command.

   This displays the Export Report form.



2. In *Report Filename*, specify a path and name for the file to be exported, or use the *Browse* function to help you specify the location.
3. Click *OK* to export the report to the specified file location.

Related Topics

Export Report Form

© 2023 Cadence Design Systems, Inc.

Recording Commands and Events in Abstract Generator

Every operation you perform during a session is recorded in a `.record` file. This allows you to rerun any session in batch mode and create user-defined macros with a minimum of manual intervention. Use the Record form to change the name of the record file, change the way the file is updated, or switch off recording altogether. The Record form controls the behavior of the record file, `abstract.record`.

By default, `abstract.record` is the name of the record file. If a file with that name already exists, the existing file is renamed to `abstract.record.`*`n`* where n is equal to 1, and the new record file is named `abstract.record`.

If the name of the record file is not changed in subsequent sessions, the `abstract.record` file is renamed to `abstract.record.`*`(n+1)* to maintain the older versions of the record file. Therefore, the greater the numeric value in the record file name, the recent (latest) that file is.

For example, if the file name is not changed in three successive sessions, the following record files are created:

- `abstract.record` (Current record file)
- `abstract.record.2` (Next oldest)
- `abstract.record.1` (Oldest)

Therefore, the `abstract.record` file is always the current record file. The other files with a numeric suffix are backups of previous record files.

1. Choose *File – Record* to start this command. This displays the Record form.

2. In *Record File*, specify the name of the record file. Use the *Browse* function to help you locate an existing file or change the current directory. If you do not change the name of the record file, it will be overwritten during the next Abstract Generator session.

© 2023 Cadence Design Systems, Inc.

3. Use the *Mode* pull-down list to select whether the commands of this session should overwrite or be appended to the record file or switch off the record function.

Note: The `.record` file does not acknowledge the selection of *File – Exit*.

Related Topics

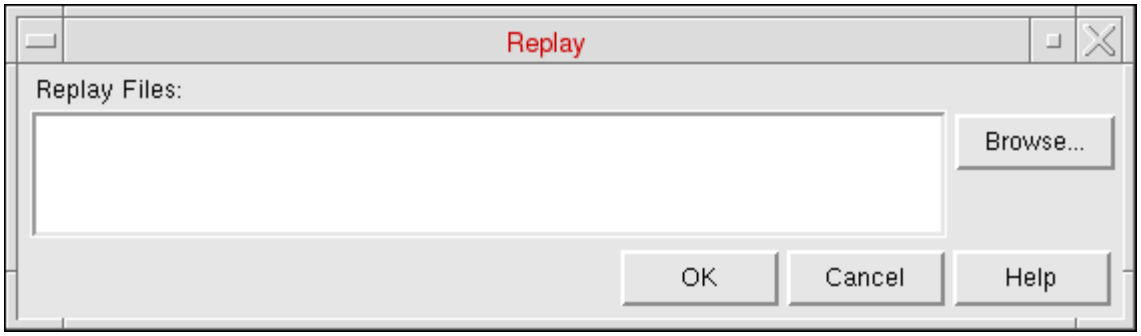
Replaying Commands in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Replaying Commands in Abstract Generator

Every operation you perform during a session is recorded in a `.record` file. This allows you to rerun any session in batch mode and create user-defined macros with a minimum of manual intervention. This file can then be replayed in either GUI or non-GUI mode.

1. Choose *File – Replay* to start this command. This displays the Replay form.



2. Specify the name of the file to be replayed in the *Replay Files* field. Use the *Browse* function to help you locate it if required.
3. Click *OK* to replay the session described in the file.

The contents of a sample replay file are shown below.

```
absSkillMode()
absSetLibrary("amsPLL")
absSelectCellFrom("pllclk25" "pllclk25")
absSetBinOption("Block" "PinsTextPinMap" "(Metal1 Metal1)(Metal2 Metal2)(Metal3
Metal3)(Metal4 Metal4)")
absAbstract()
absSetOption("ExportLEFVersion" "5.5")
absExportLEF()
absExit()
```

When replaying a session, the record function is automatically turned off.

Related Topics

Recording Commands and Events in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Conversion of Common Lisp Files to SKILL or TCL Format

This section describes how to convert any Abstract Generator replay files you have from common-lisp format into SKILL or TCL format. As SKILL is used extensively in the Virtuoso Studio design environment, Cadence recommends that you convert any replay files from common-lisp to SKILL format. To do this, use the Abstract Common Lisp Converter, `absclconv`.

You should use `absclconv` ***to convert only Abstract Generator common-lisp files. It is not designed as a generic common-lisp conversion utility.***

The command accepts the following arguments.

```
absclconv
     { -tcl | -skill }
     clFile
     newFile
```

These arguments are described in detail below.

`-tcl | -skill`	Specifies the format of the file to be generated from the common-lisp file.
`clFile`	Specifies the common-lisp file to be converted.
`newFile`	Specifies the name of the file to be created in the new format.

For ease of use, the following shortcuts are provided.

- To convert a common-lisp file directly to SKILL format, use the `abscl2il` link.

  `abscl2il` *`clFile ilFile`*

  This creates a new SKILL file, *`ilFile`*, based on the common-lisp file *`clFile`*.

- To convert a common-lisp file directly to Tcl format, use the `abscl2tcl` link.

  `abscl2tcl` *`clFile tclFile`*

  This creates a new Tcl file, *`tclFile`*, based on the common-lisp file *`clFile`*.

In all cases, if a file with the same name already exists, it is overwritten.

© 2023 Cadence Design Systems, Inc.

`absclconv` preserves comments and blank lines, so a command that is commented out in the common-lisp version remains commented-out in the SKILL and Tcl versions. (The comments are displayed differently in SKILL and Tcl.)

The example below shows how a common-lisp command converts to Tcl and SKILL formats.

Common-Lisp	Tcl	SKILL
`(abs:Select "param1" "param2")`	`absSelect "param1" "param2"`	`absSelect ("param1" "param2")`

Related Topics

Abstract Generator Log, Replay, and Record Files

© 2023 Cadence Design Systems, Inc.

Options and Status Files in Abstract Generator

Abstract Generator saves the options file (`.abstract.options`) in the `.abstract/<library name>` directory in your current working directory. It no longer saves the file in the current design library. The `.abstract` directory is created in your current working directory when Abstract Generator runs.

Consider, for example, that you are working with the design library `myLib` and your current working directory is `/home/user1`. In this case, Abstract Generator saves the options file in `/home/user1/.abstract/myLib/.abstract.options`. Abstract Generator also saves the existing options file as `.abstract.options.prev` in the same directory where the `.abstract.options` file is saved.

Abstract Generator loads the options file in a specific sequence.

- It first loads the options file (`.abstract.options`) from the `<current working directory>/.abstract/<library name>` directory.
- If not available in the above location, it reads the file first from the attached technology library directory and then from the design library directory.

If the same options are available in the next file loaded, the options are overwritten accordingly.

Abstract Generator saves the status file (`.abstract.status`) in the `.abstract/<library name>/cellStatus` directory in your current working directory. It no longer saves the file in the cell directory. Information about each cell is stored in a separate status file. The status file stores the following information:

- Name of the active cell in the selected bin
- View statuses (Pins, Abstract, or Extract View) of all cells in the selected bin
- Status of the cell when it is moved from one bin to another

The status file is read when a library is loaded. The file is updated when the pins, extract, and abstract steps are run, and also when the library is closed.

```
cUiSelectCellP {test1}
cUiSetViewStatus {test1} {layout} {1}
cUiSetViewStatus {test1} {logical} {0}
cUiSetViewStatus {test1} {abstract.pin} {3}
cUiSetViewStatus {test1} {abstract.ext} {1}
cUiSetViewStatus {test1} {abstract} {3}
```


© 2023 Cadence Design Systems, Inc.

For backward compatibility, Abstract Generator copies information from the `.abstract.status` file at the cell level to the `.abstract.status.cellName` file at the user directory level. Any further changes are done in the `.abstract.status.cellName` file.

Related Topics

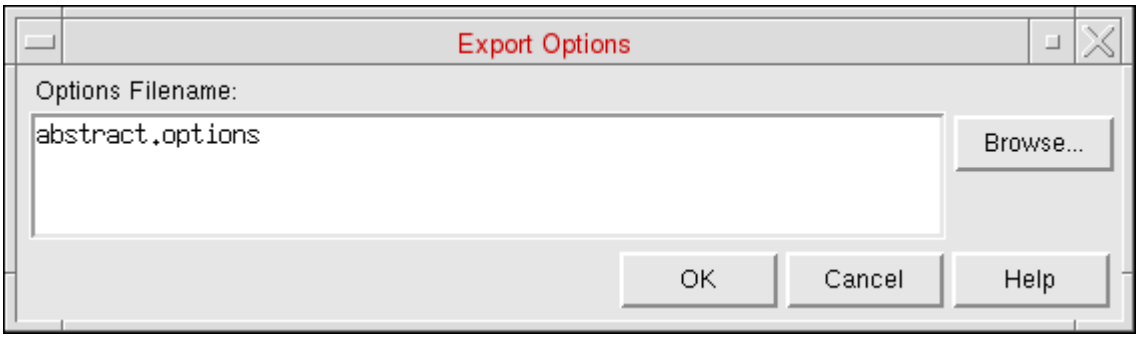
Conversion of Common Lisp Files to SKILL or TCL Format

© 2023 Cadence Design Systems, Inc.

Exporting Options File in Abstract Generator

The options file records settings for both global and bin options as well as default settings for various other commands. You can use the options file to import these settings the next time you use Abstract Generator, or to restore them later in the current session after completing any experiments with alternative settings.

1. Choose *File – Export – Options* to display the Export Options form. This command saves the export option settings for the current session.



2. In the *Options Filename* field, specify a path and name for the file, or use the *Browse* function to help you specify the location.
3. Click *OK* to export the options to the specified location.

Related Topics

Export Options Form

© 2023 Cadence Design Systems, Inc.

Guidelines for Optimizing Abstract Generator Performance

Some abstract generation functions can adversely affect the performance of the tool and should be run only when necessary. Keep the following guidelines in mind while using the standalone Abstract Generator interface:

- Do not extract signal and power nets for blocks or IO cells unless you are sure you need to do so. For example, you must extract signal nets if you want to perform antenna extraction, while you might need to extract power nets if you want to create ring pins.
- If you do need to extract signal or power nets,
  - Do not create pins on a given layer unless it is necessary. For example, do not create pins on polysilicon, diffusion, or contact layers, except where there are dominant cut-layer spacings.
  - Define the `Poly` layer as `Poly and Oxide` and `Oxide` layer as `Oxide andnot Poly` in the geometry specifications under the *Extract signal nets* and *Extract power nets* sections in the Extract step.
  - You may not require the *Maximum depth* value to be 32 as per your design scenario under the *Extract Limitations* section in the Extract step. Limit the extraction by the hierarchy depth.
  - Ensure in the Abstract step that not all of the extracted geometry is turned into pins. If you do not do this, Abstract Generator must cut around each pin, which increases the time taken to generate the blockage model. You can avoid this for power pins by creating ring or boundary pins and for signal pins by creating boundary pins.
- Turn off the *Fracture pins* and *Fracture blockages* options if you observe that Abstract Generator is taking a lot of time in the Abstract step.
- Do not create detailed blockages on contact or via layers for large blocks. The large number of contact obstructions slows down the Abstract step.
- Do not run grid analysis for blocks or IO cells.
- If blocks or IO cells have pins or obstructions with 45-degree shapes, you must either ensure that they are removed from the abstract (for example, if they are not in pins, then the cover blockage will hide them) or consider increasing the stair-step width.
- If pins and extract cellviews are not required by the user, the `QuickAbstract` option can be enabled to improve the performance of Abstract Generator by 30 percent on small- and mid-size designs. When the `QuickAbstract` option is set to true, then Abstract Generator will not generate the intermediate cellviews that is Pin and Extract

© 2023 Cadence Design Systems, Inc.

cellviews. Only the Abstract cellview will be generated. By default, the `QuickAbstract` option is set to false.

- Take care when creating geometry specifications to act on pins on the `prBoundary` layer. This layer can feature thousands of shapes and an incorrect geometry specification can cause these to be merged into a single rectangle for each pin.
- Ensure to correctly define the *Gate/Drain* region under the *Antenna* tab in the Abstract step for antenna calculations. If the *Use different layer assignments for antenna calculations only* option is turned on, define the `Poly` layer as `Poly` and `Oxide` layer as `Oxide andnot Poly` in the geometry specifications.
- Limit the *Geometry search depth* to a lower value if required, under the *Text* tab in the Pins step.
- If Abstract Generator is running out of memory while extracting nets on large designs and you get the following message:

  ```
  *WARNING* dbDeleteShape: A memory allocation for 1073741824 bytes
  failed. The process heap is 511MB, and OpenAccess is using 1405MB.
  The available system memory is 991MB, and physical system memory
  is 7857MB.
  ```

  Set the following option in the Abstract Generator console or script files.

  ```
  absSetOption("OptimizeBigDesignMemory" "true")
  ```

  This option takes care of memory issues that arise on the big block designs in Abstract Generator. By default, this option is set to `false`. To optimize the memory for big designs, set this option to `true`. Also, ensure that the abstract executable is run in 64-bit mode.
- If extraction is aborted because of an excessive number of shapes while extracting the power nets on large designs, increase the limit of shapes to be extracted using the following option. The default value is `10000`.

  ```
  absSetOption( "ExtractShapeLimit" "150000")
  ```

Related Topics

Pins Step in Standalone Abstract Generation

Extract Step in Standalone Abstract Generation

Abstract Step in Standalone Abstract Generation

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

3

Pins Step in Standalone Abstract Generation

The Pins step describes the first step in the abstract generation process. Usually, no pins are explicitly defined in the layout view, so they have to be generated before you can continue the abstract generation process. Abstract Generator derives pins from the text labels in the layout view and places the locations at the text origins.

- If there is a logical view present, Abstract Generator only creates pins that match logical view terminals.
- Abstract Generator copies the placement status of the pins in the layout view to the abstract view. The supported placement states are `locked`, `placed`, `fixed`, `none`, and `unplaced`.
- Abstract Generator automatically detects shapes placed in width spacing pattern (WSP) regions and ignores these shapes while creating pins under labels and PR boundaries.

Abstract Generator provides a number of options that support a variety of layout and text mapping conventions. To check options for mapping and for controlling the generation of pins, see Specifying Pin Mappings for Abstract Generation.

Abstract Generator performs the following functions during the Pins step:

- Maps text labels to terminal names and creates terminals
- Creates physical pin shapes corresponding to the geometry overlapping the text labels
- Creates the place-and-route boundary

You can use the Running step Pins form to view and modify the bin options relevant to the Pins step.

To run the Pins step:

1. Select one or more cells in the main window and click the *Pins* button. Alternatively, select *Flow – Pins* command.

© 2023 Cadence Design Systems, Inc.

This opens the Running step Pins form. There are four tabs on the Running step Pins form: *Map*, *Text*, *Boundary*, and *Blocks.* You can use the radio buttons on the left to navigate to the tab pages for the different steps and bins you are using in the Running step Pins form.

2. When you are satisfied with the options settings, click *Run* to run the step.

The *Percent Complete* message box reports the progress of the step. When the Pins step is complete, Abstract Generator places a symbol against the cell in the *Cell* pane to indicate the status of the Pins view that was created.

Related Topics

Extract Step in Standalone Abstract Generation

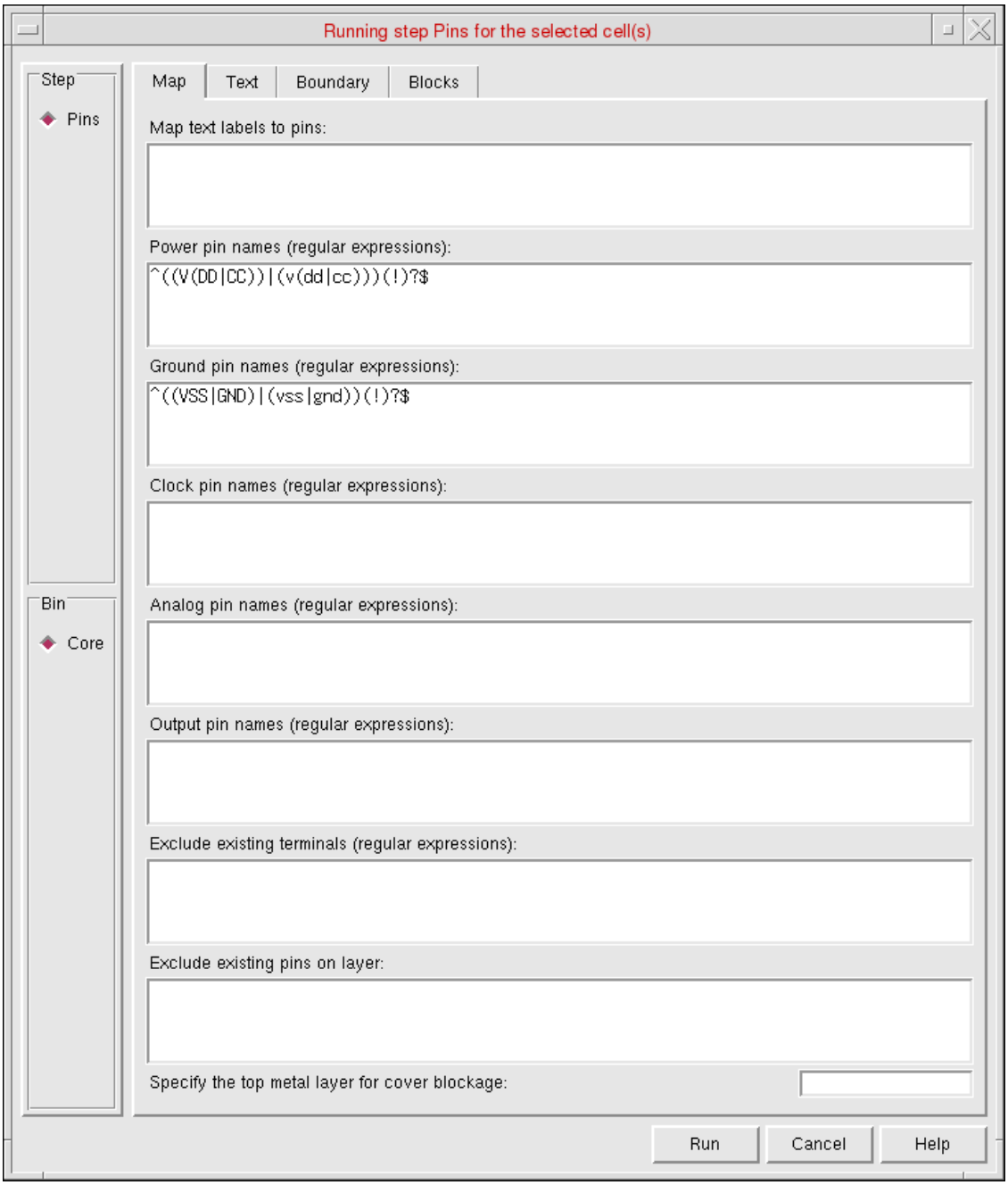
Abstract Step in Standalone Abstract Generation

© 2023 Cadence Design Systems, Inc.

Specifying Pin Mappings for Abstract Generation

To specify pin mapping settings in Standalone Abstract Generator:

1. Choose *Flow – Pins* or click the *Pins* icon.



The Running step Pins form is displayed. The *Map* tab is the first tab in the form.

© 2023 Cadence Design Systems, Inc.

On the *Map* tab, you can specify the text labels to be used for creating starting pin shapes for each of the nets to be extracted. You can also specify the pins that are power, ground, clock, analog, or output pins.

2. In the *Map text labels to pins* field, specify the text labels in the layout view that are to be mapped to the pins on nets in the pins view. Specify the layer-purpose pairs based on which you want Abstract Generator to search for geometry of any given layer-purpose pair with a text label.
3. Specify the pin names in the *Power pin names*, *Ground pin names*, *Clock pin names*, *Analog pin names*, and *Output pin names* fields. Abstract Generator identifies the respective pin names based on the label strings in the abstract.
4. In the *Exclude existing terminals* field, specify a list of existing terminals that you want Abstract Generator to ignore.
5. In the *Exclude existing pins on layers* field, specify a list of layers on which you want Abstract Generator to ignore the existing pins.
6. In the *Specify the top metal layer for cover blockage* field, specify the metal layer to cover blockages.
7. Click *Run*.

The *Percent Complete* message box reports the progress of the step.

Related Topics

Running Step Pins Form

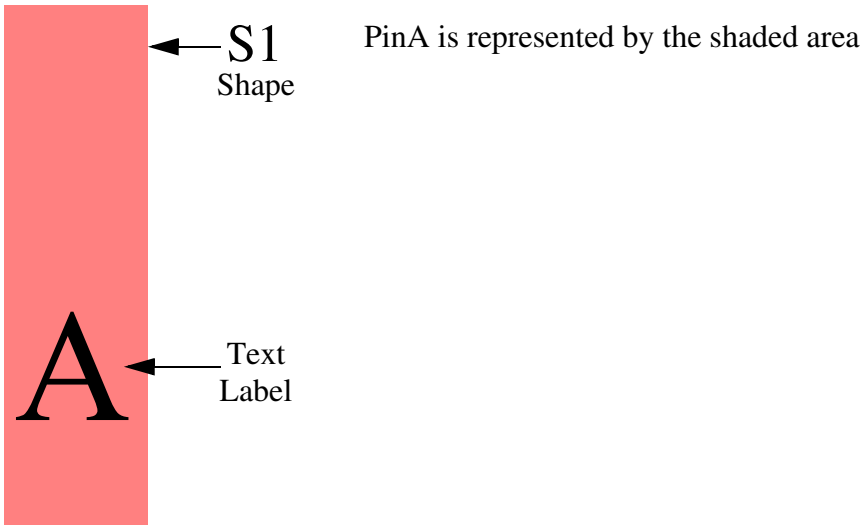
Text Labels-to-Pins Mapping

Troubleshoot Pin Issues

© 2023 Cadence Design Systems, Inc.

Text Labels-to-Pins Mapping

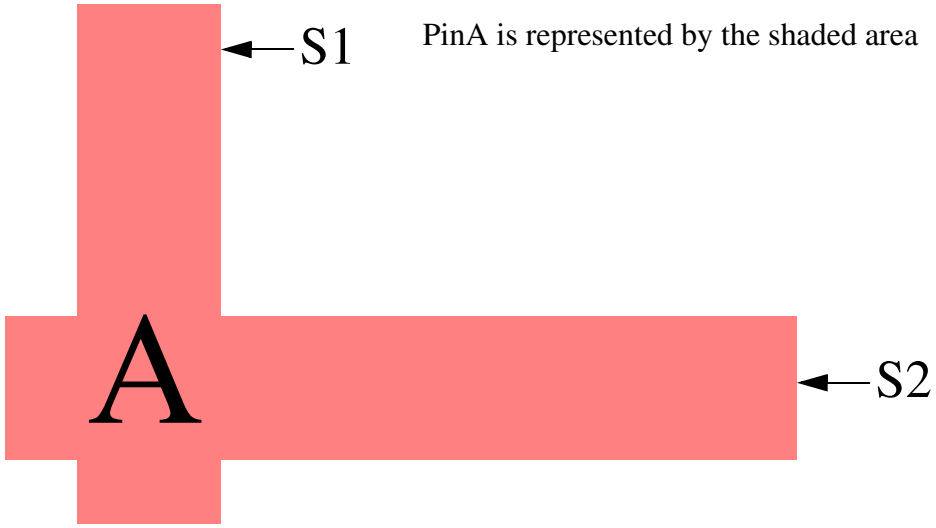
In the *Map text labels to pins* field on the *Map* tab of the Running step Pins form, you can specify the text labels in the layout view that are to be mapped to the pins on nets in the pins view. For each statement, Abstract Generator first identifies all the text labels on the specified `textLPP`. For each occurrence of a text label that overlaps a shape on the first `geomLPP`, it creates a pin of the same size as the shape on the metal layer overlapping the text label. This is depicted in the figure below. The text label `A` on `metal1 pin` overlaps the shape `S1` on `metal1 drawing`. So Abstract Generator creates the vertical `PinA`, which is of the same size as `S1`.



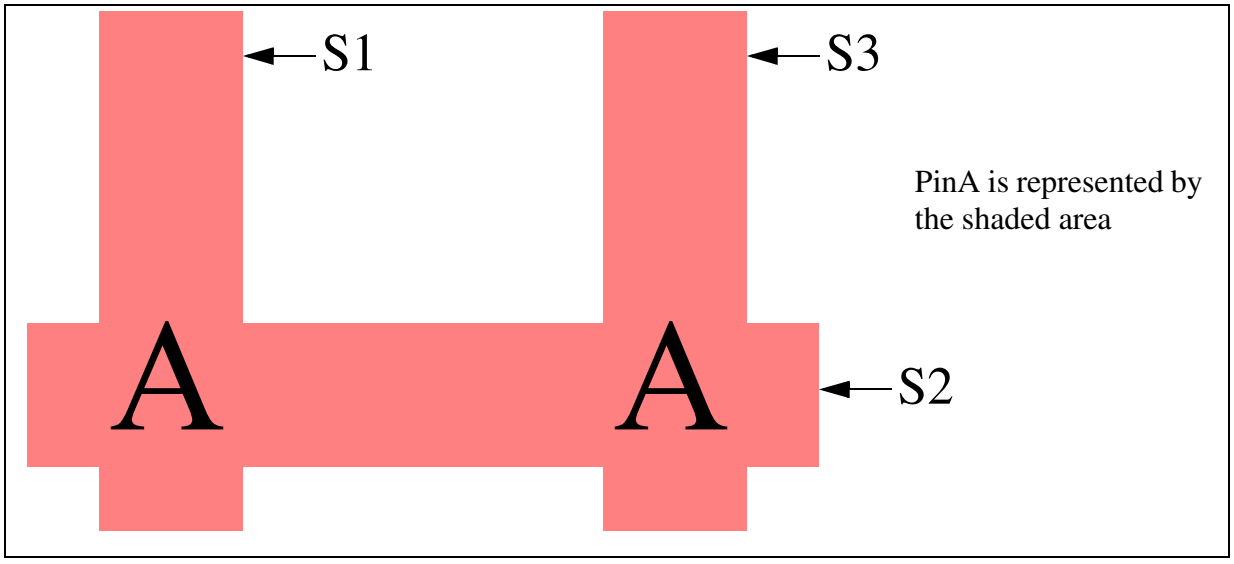
In case the text label overlaps more than one shape, Abstract Generator creates a single merged pin for the overlapped shapes. As shown in the figure below, the text label `A` overlaps

© 2023 Cadence Design Systems, Inc.

two intersecting shapes `S1` and `S2`. In this case, Abstract Generator merges the vertical and horizontal pin shapes and creates the single L-shaped `PinA`, which is cornered at left.



Furthermore, Abstract Generator creates pins for all the text labels with the same name, irrespective of whether a shape has already been made into a pin by a previously evaluated text label. It also merges the overlapping pin shapes generated from the text labels with the same name. This is illustrated in the figure below where another text label with the same name `A` overlaps the shapes `S2` and `s3`. Even though Abstract Generator already created a pin on `S2` from the first text label `A`, it creates another pin shape from the overlap of the second text label `A` with the shapes `S2` and `S3`. The overlapping pin shapes in this situation result in a single merged U-shaped `PinA`.



© 2023 Cadence Design Systems, Inc.

Abstract Generator proceeds in the similar manner for all other text labels on `textLPP`. The order in which Abstract Generator analyzes shapes is from the first specified `geomLPP` to the next.

Consider the following examples as the text that can be specified in the *Map text labels to pins* field:

- `(textLayer (metal1 drawing) (metal2 drawing))`

  This means that text labels on `textLayer` for all purposes except the ignored purpose(s) are mapped first to shapes on layer `metal1` purpose `drawing` and then to shapes on layer-purpose pair `metal2 drawing`.

- `(textLayer metal1) (textLayer metal2)`

  This means that labels on `textLayer` for all purposes except the ignored purpose(s) are mapped first to shapes on `metal1` excluding the ignored purpose(s). Then, in a second pass, Abstract Generator maps labels on `textLayer` to shapes on `metal2` excluding the ignored purpose(s). To do this, Abstract Generator takes the first mapping, `(textLayer metal1)`, and creates appropriate pins on the `metal1` layer. If there are text labels overlapping `metal2` shapes, they are not matched by this rule and Abstract Generator issues a number of ABS-12006 warnings informing you about labels that could not be mapped to pins. These labels are picked up in the second pass.

Unless there are terminals already present, you must specify a value in the *Map text labels to pins* field to generate a valid pins view. If you leave the field empty, Abstract Generator does not generate pins in the abstract view and the warning message ABS-12004 is displayed. To generate these pins, specify valid values in the *Map text labels to pins field*, and rerun the Pins step.

Pins Step for Routed Blocks

You do not need to specify a value in the *Map text labels to pins* field when running the Pins step on routed blocks. Routed blocks already have terminals, so Abstract Generator does not create any new terminals. If the existing terminals have physical pins, as is usually the case with signal nets, these are retained in the Pins step. If these pins are not on valid LEF layers, Abstract Generator deletes these pins after generating the ABS-12023 message.

Otherwise, all the routed shapes attached to the net are turned into pins on the net. This is the case with power nets, which usually have terminals but no physical pins.

Related Topics

Specifying Pin Mappings for Abstract Generation

© 2023 Cadence Design Systems, Inc.

Regular Expressions for Creating Pin Names

Pin Names

Troubleshoot Pin Issues

© 2023 Cadence Design Systems, Inc.

Regular Expressions for Creating Pin Names

All fields in which you specify power, ground, output, analog, and clock pin names accept regular expressions. The pin name in question is formed from the subset of the label text that matches the regular expression you enter.

Consider the following examples:

- To create power pins *only* from labels that match the string `VDD`, specify the following power pin name regular expression:

  ```
  ^VDD$
  ```

  The `^` matches the beginning of a string; the `$` matches the end of a string. Therefore, only strings containing the `VDD` match the regular expression.

- To create a pin name that is identical to a label text containing the string `VDD` anywhere, specify `.*VDD.*` as the regular expression.

  In regular expressions, `.*` matches any sequence of characters.

  Note: Abstract Generator does not check the regular expression you enter against any syntax rules.

Related Topics

Specifying Pin Mappings for Abstract Generation

Text Labels-to-Pins Mapping

Troubleshoot Pin Issues

© 2023 Cadence Design Systems, Inc.

Pin Names

You can specify the pin names in the *Power pin names*, *Ground pin names*, *Clock pin names*, *Analog pin names*, and *Output pin names* fields of the Running step Pins form. Abstract Generator identifies the respective pin names based on the label strings in the abstract. You can specify a list of possible names for the pins in the form of regular expressions, each separated by a space, as shown in the figure below.

By default, the *Clock pin names*, *Analog pin names*, and *Output pin names* fields are empty.

When Abstract Generator encounters a label that matches one of the regular expressions, it sets properties as given in the table below. In addition, Abstract Generator adds attributes for the pin in question to the LEF definition.

Pin	Terminal Direction	Terminal Type	Attribute Added for LEF PIN in the LEF File
Power pin	`inputOutput`	`power`	`USE POWER`
Ground pin	`inputOutput`	`ground`	`USE GROUND`
Clock pin	`input`	`clock`	`USE CLOCK`
Analog pin	`input`	`analog`	`USE ANALOG`
Output pin	`output`	`signal`	`USE OUTPUT`

If there is a logical view present for the cell being processed, Abstract Generator checks whether a given terminal has a matching terminal in the logical view. If it does, the terminal direction is adopted from the terminal in the logical view. Otherwise, Abstract Generator uses the value specified in the *Output pin names* field to determine the output pins.

Related Topics

Specifying Pin Mappings for Abstract Generation

Regular Expressions for Creating Pin Names

© 2023 Cadence Design Systems, Inc.

Text Labels-to-Pins Mapping

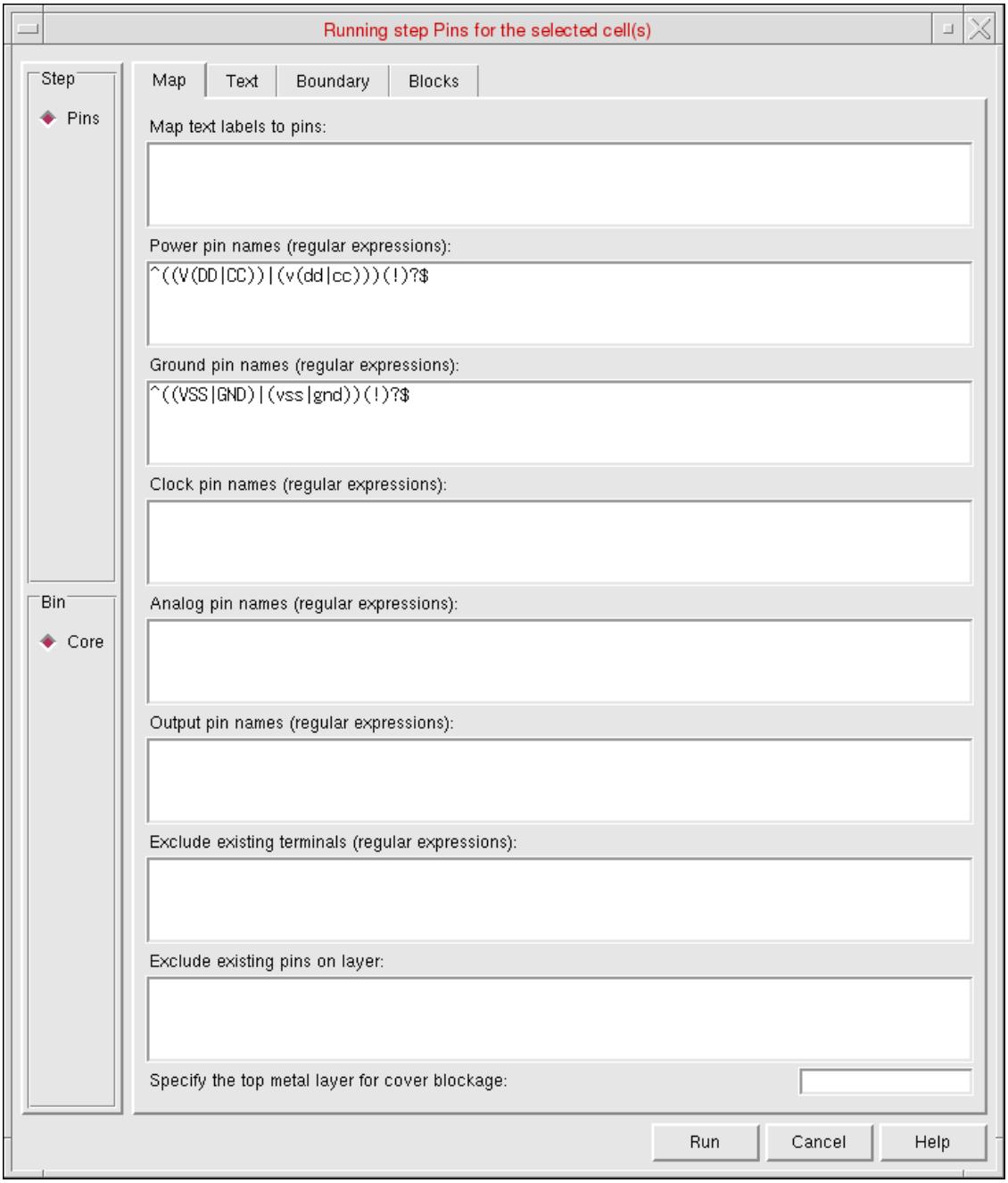
Troubleshoot Pin Issues

© 2023 Cadence Design Systems, Inc.

Customizing Text Labels for Abstract Generation

To customize text labels in the Standalone Abstract Generator:

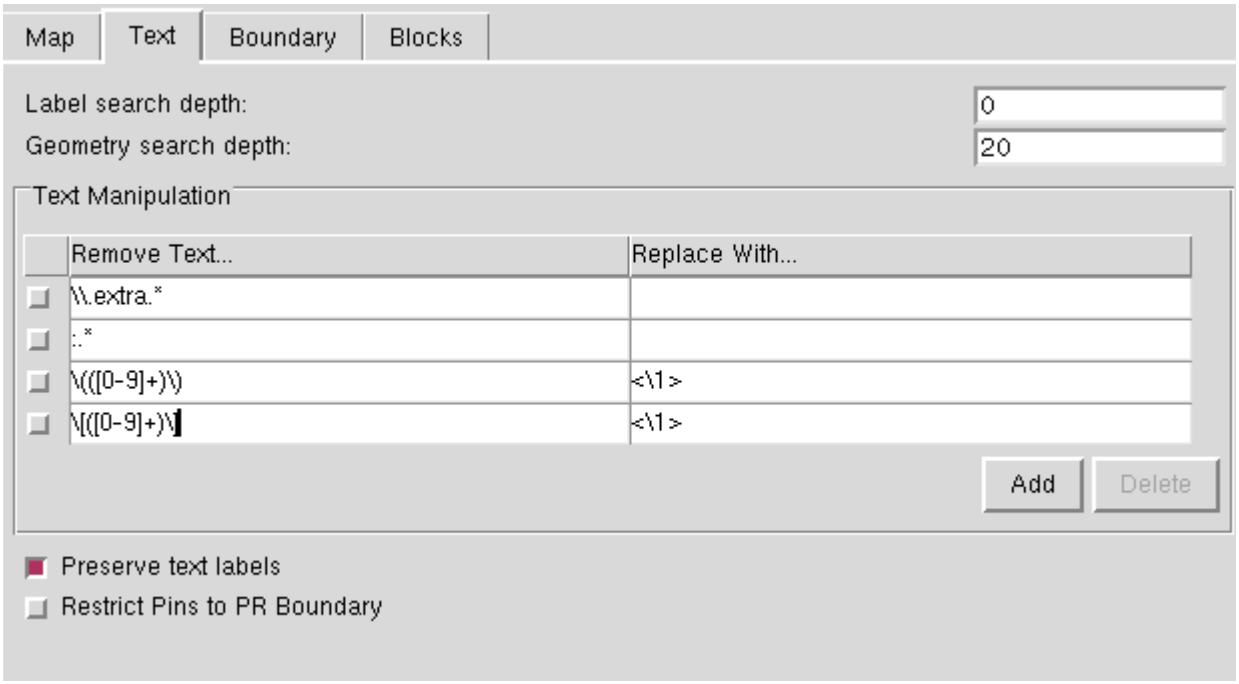
1. Select one or more cells in the main window and choose *Flow – Pins*, or click the *Pins* icon.



© 2023 Cadence Design Systems, Inc.

The Running step Pins form is displayed.

2. Select the *Text* tab.



3. In the *Label search depth* field, specify a value to control how far down the design hierarchy does Abstract Generator search for text labels during text-to-pin mapping.
4. In the *Geometry search depth* field, specify a value to control how far down the design hierarchy does Abstract Generator search for metal underneath a text label.
5. In the *Text Manipulation* table, specify the expression to be replaced in the *Remove Text* column and the text to be replaced with in the *Replace With* column.
6. Enter the following in the *Text Manipulation* fields to change parenthesis, `()`, to angle brackets, `<>`, for the label `(Y)`,

Remove Text...	Replace with...	Result
`\(`	`<`	`<Y)`
`\)`	`>`	`<Y>`

7. Click *Add* to add a new row to the table.
8. Click *Delete* to delete the selected rows from the table.

© 2023 Cadence Design Systems, Inc.

9. Select *Preserve text labels* to retain all existing pins in the layout, along with their labels, irrespective of whether the pins were created by Abstract Generator during the Abstract step.
10. Select *Restrict Pins to PR Boundary* to control the geometry of N-Well pins created during the abstract generation process.
11. Click *Run*.

    The *Percent Complete* message box reports the progress of the step.

The customized text labels are generated in the layout.

Set the PinsFromTextForExistingPins option to `t` to generate additional pins from labels, even if the corresponding net has existing pins in layout.

Related Topics

Running Step Pins Form

Text Labels-to-Pins Mapping

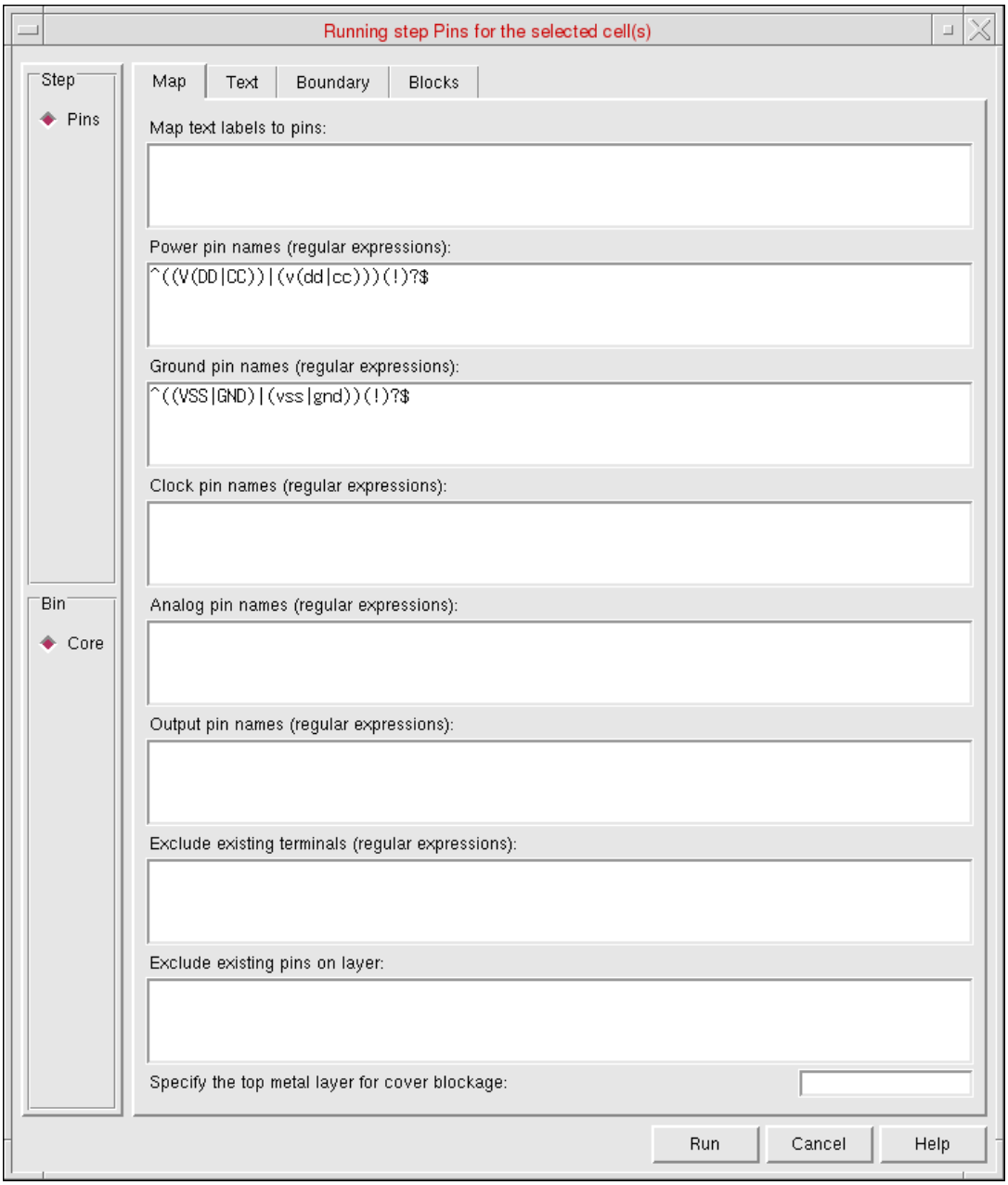
Troubleshoot Pin Issues

© 2023 Cadence Design Systems, Inc.

Setting PR Boundary Creation Options in Standalone Abstract Generation

To control the calculation of the place-and-route (PR) boundary in Running Step Pins form:

1. Select one or more cells in the main window and choose *Flow – Pins,* or click the *Pins* icon.



© 2023 Cadence Design Systems, Inc.

The Running step Pins form is displayed.

2. Click the *Boundary* tab.

3. In the *Create boundary* cyclic field, you can specify whether Abstract Generator is to create a PR boundary by choosing one of the values from *always*, *as needed*, and *off*.

   Abstract Generator creates an OA PR boundary during the Pins step itself and propagates it during the Extract and Abstract step.

4. In the *Using geometry on layers* field, specify the layers to be used to calculate the PR boundary

5. In the *Adjust Boundary By* field, you can increase or decrease the size of an existing or calculated PR boundary on one or more sides. To do this, specify the distance to be added to each boundary edge.

6. In the *Fix Boundary To* field, you can fix individual cell boundary edges to an absolute value. These values are applied to existing and calculated PR boundaries.

7. Click *Run*.

   The *Percent Complete* message box reports the progress of the step.

© 2023 Cadence Design Systems, Inc.

The PR boundary is created in the layout. The PR boundary shapes, which have the `placementClass` property set, are considered by Abstract Generator to be *placement regions*. The `placementClass` property indicates that instances with a matching `placementClass` can be placed in the same placement regions.

Abstract Generator ignores placement regions while determining whether a PR boundary exists. To enable Abstract Generator to correctly recognize a PR boundary, the instance must not have the `placementClass` property set.

Related Topics

Running Step Pins Form

Customizing Text Labels for Abstract Generation

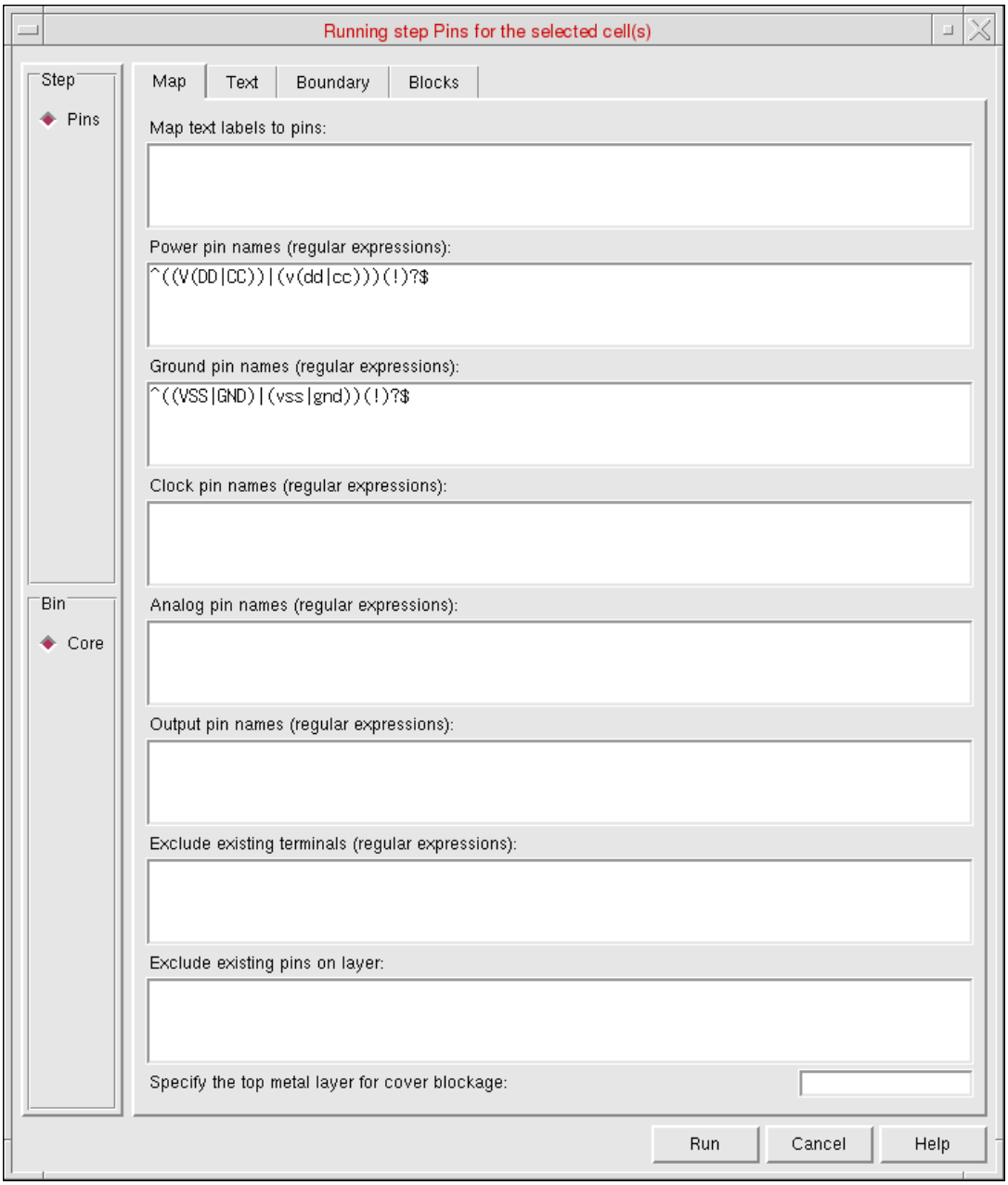
Troubleshoot Pin Issues

© 2023 Cadence Design Systems, Inc.

Customizing Abstracts of Routed Blocks in Standalone Abstract Generator

To customize abstracts of routed blocks in Standalone Abstract Generator:

1. Select one or more cells in the main window and choose *Flow – Pins,* or click the *Pins* icon.



© 2023 Cadence Design Systems, Inc.

The Running step Pins form is displayed.

2. Click the *Blocks* tab.

3. Select the *Preserve local blockages in routed blocks* field to control how Abstract Generator processes local blockages defined in the *Blockage* section in DEF 5.8. Local blockages do not have the `+PUSHDOWN`, `+FILL`, or `+SLOT` modifiers.
4. Select the *Create power pins from routing* field to control whether pins are created from the routing defined in the `SPECIALNETS` section of the DEF file.

   When checked, Abstract Generator creates pins from the routing defined in the `SPECIALNETS` section for the layers specified in the *Using geometry on layers* field. It does this even if there are pins already defined in the *Pins* section.
5. Click *Run*.

   The *Percent Complete* message box reports the progress of the step.

Related Topics

Running Step Pins Form

Troubleshoot Pin Issues

© 2023 Cadence Design Systems, Inc.

N-Well and P-Well Pins Creation

Abstract Generator supports the creation of N-Well and P-Well pins. During abstract generation, the tool recognizes the text labels for N-Well and P-Well shapes on the layout and creates corresponding physical pins.

N-Well Pins

To create N-Well pins, during the Pins step, you need to:

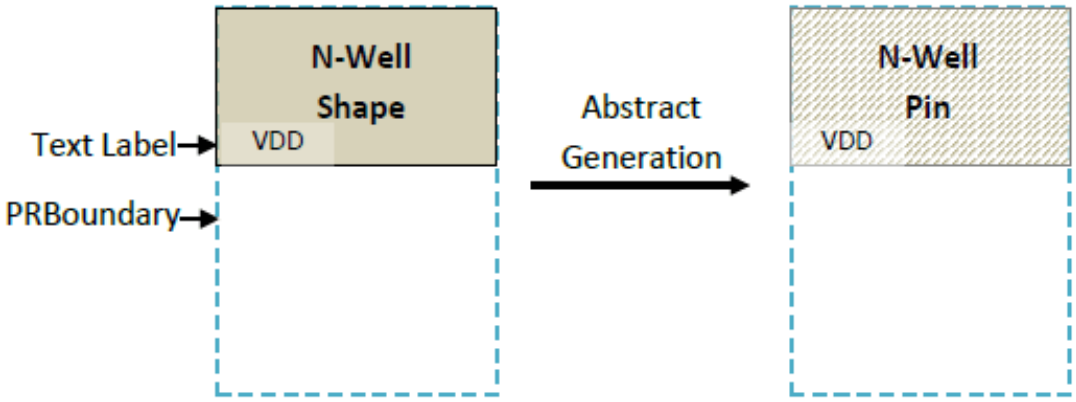
- Define the geometric specification of N-Well shapes.

  Example: (Nwell (Nwell drawing))

- Create text labels for the N-Well pins.

During abstract generation, the tool identifies all text labels on the layout design. Then, it recognizes the shapes overlapping each text label and creates pins of the same size as the overlapping shape.

Example:

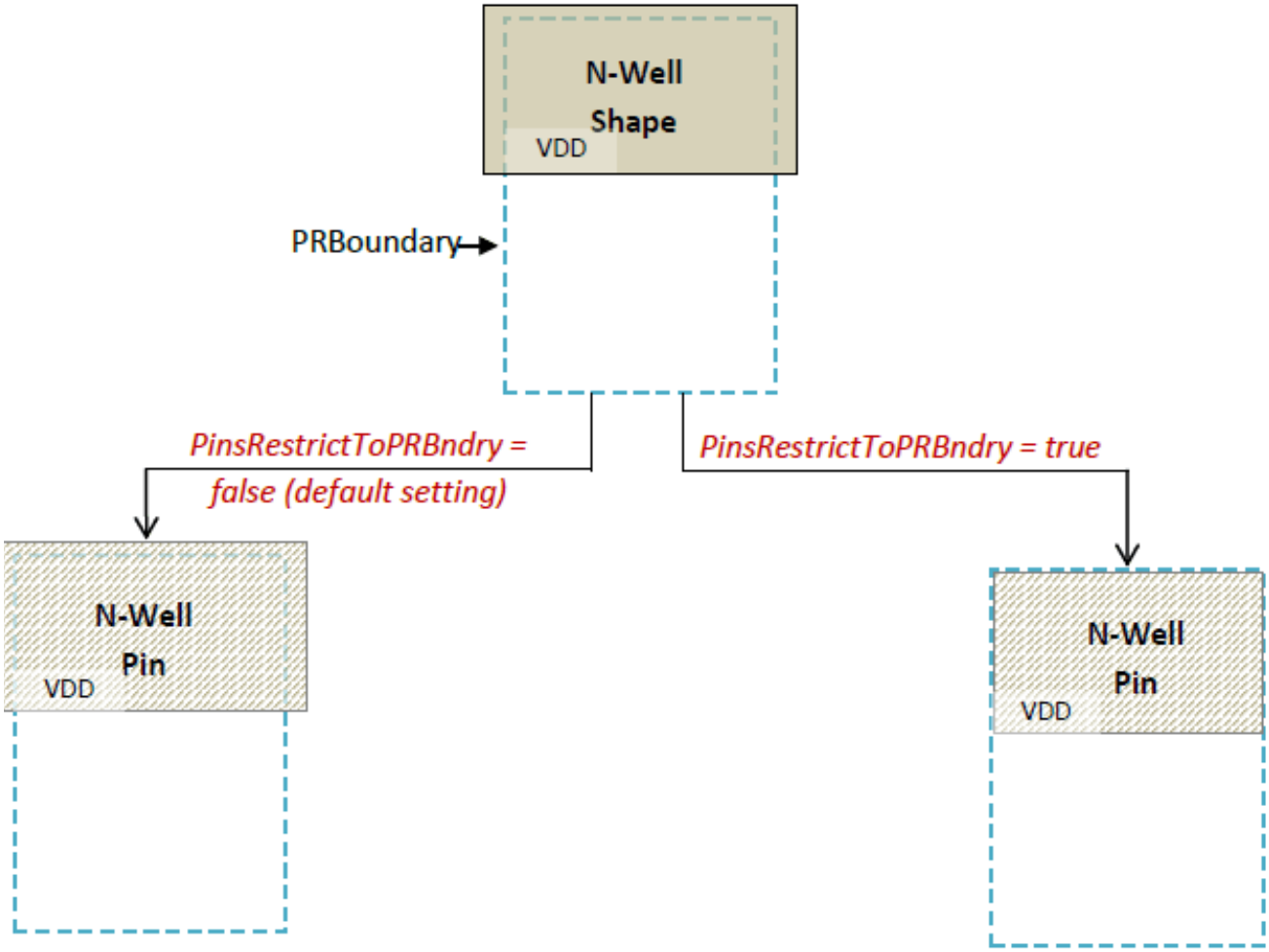


Note: If there is no Nwell shape overlapping the label, then no pin is created.

For some designs, the N-Well shape boundary may exceed the PRBoundary. By default, Abstract Generator will create N-Well pins of the same size as the N-Well shape.

© 2023 Cadence Design Systems, Inc.

The `PinsRestrictToPRBndry` option enables you to limit the N-Well pin size to the PR boundary. The default value is false.



P-Well Pins

To create P-Well pins, during the Pins step, you need to:

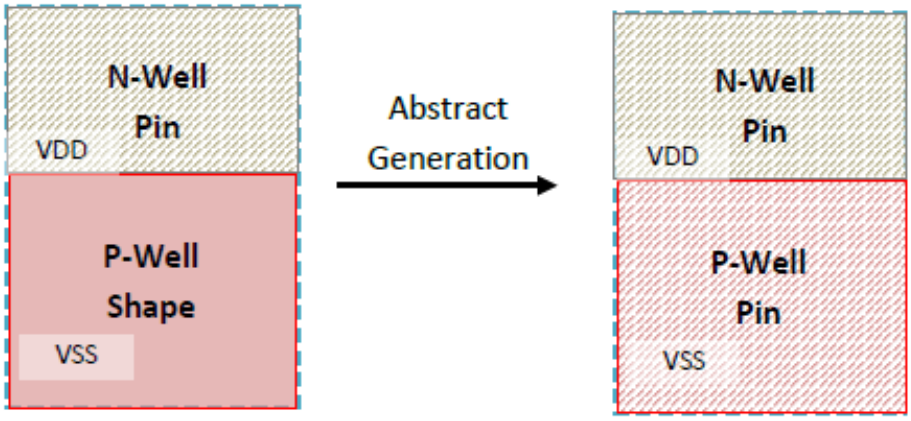
- Define the geometric specification of the P-Well shapes.

  Example: (Pwell (Pwell drawing))

- Create text labels for the P-Well pins.

© 2023 Cadence Design Systems, Inc.

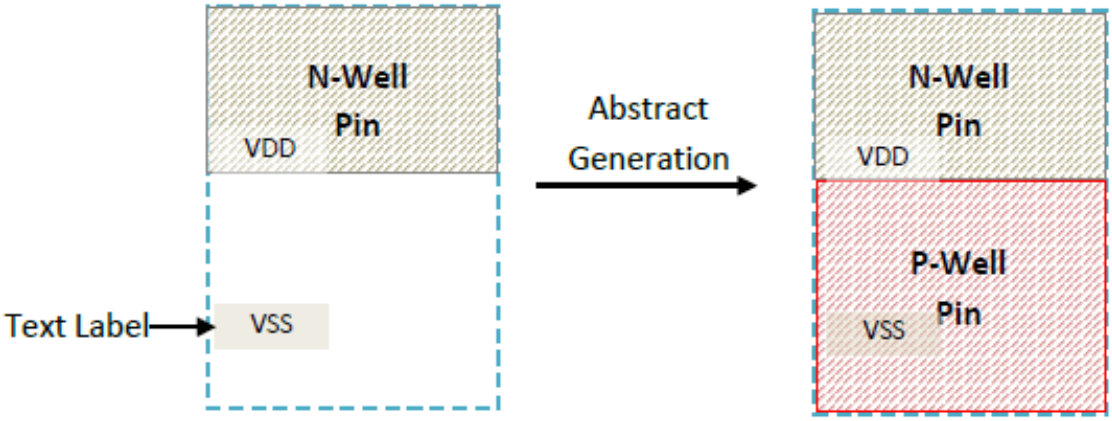
During abstract generation, the tool identifies all text labels on the layout design. Then, it recognizes the shapes overlapping each text label and creates P-Well pins of the size of the overlapping P-Well shapes.



If there are no Pwell shapes overlapping a text label, then the size of the P-Well pin is calculated as follows:

```
Area (P-Well Pin) = Area (PRBoundary) - Area (N-Well Pin)
```

The Pwell pin is placed adjoining the Nwell pin.



Multiple N-Well and P-Well Pins

To create multiple N-Well or P-Well pins, define the geometric specification of the shapes and then create text labels for the pins.

Examples:

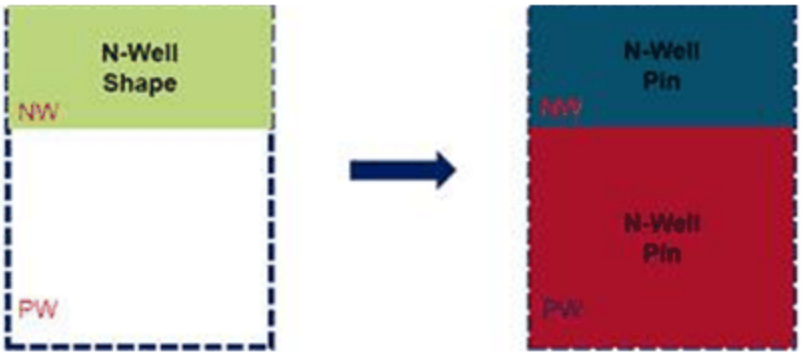
© 2023 Cadence Design Systems, Inc.

Defining the geometric specification of the N-Well shapes: (Nwell (Nwell drawing))

Defining the geometric specification of the P-Well shapes: (Pwell (Pwell drawing))

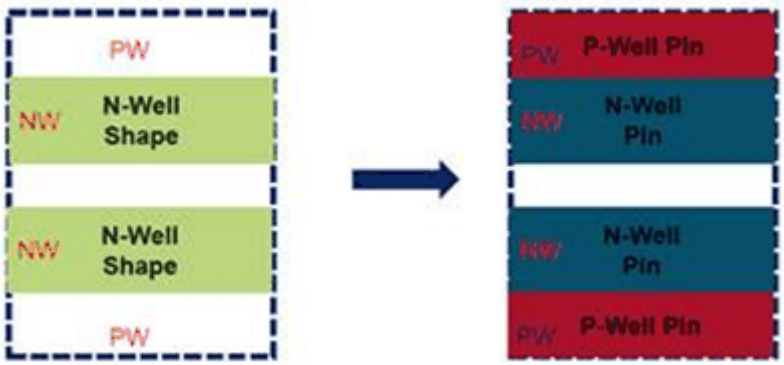
If there are no P-Well shapes under the P-Well label, then the P-Well pin are is calculated using the following formula:

Pwellpin = TotalPrBoundary - NwellPin



© 2023 Cadence Design Systems, Inc.

If an N-Well shape cuts the PrBoundary into multiple regions, then multiple P-Well pins are created.



P-well pins are created only in the regions that have the P-well labels. Notice that a P-Well pin is not created in the middle region of the above example.

If there are multiple P-Well labels at the same region, then a P-Well pin is created only for the first label, and a warning is displayed for the remaining labels.

Related Topics

Specifying Layers and Geometries for Extraction

Customizing Abstracts of Routed Blocks in Standalone Abstract Generator

© 2023 Cadence Design Systems, Inc.

Troubleshoot Pin Issues

You can face the following issues while using the Pins step in the abstract generation process.

ABS-12002

Message: Cell *name*: No terminals have been created. Use the *Map text labels to pins* option in the *Map* tab of the *Pins* step to define cell pins. See Text Labels-to-Pins Mapping.

ABS-12004

Message: Cell *name*: The abstract generator could not generate pins because the *Map text labels to pins* field did not contain valid values. You can proceed with the Extract step. However, to generate pins in the abstract view, specify valid values in the *Map text labels to pins* field and rerun the Pins step. See Text Labels-to-Pins Mapping.

ABS-12006

Message: Cell *name*: No pin will be created for the text label '*label*' at (*x*,*y*) because no overlapping geometry exists for this text label. The abstract generator creates pins corresponding to geometries that overlap the text labels specified in the *Map text labels to pins* field on the *Map* tab. Therefore, while updating this field with pin mapping specification, ensure that geometries exist below the text labels specified. You can also check to ensure that the GDSII layout data has been imported into the abstract generator on correct layers.

Description: Abstract Generator creates pins corresponding to geometries that overlap the text labels specified in the *Map text labels to pins* field of the Pins step. This error message is generated when Abstract Generator does not find any shape overlapping the text label specified in the *Map text labels to pins* field.

ABS-12007

© 2023 Cadence Design Systems, Inc.

Message: Cell *name*: A pin has already been created on the net *name*, which conflicts with the label *name* at (*x*,*y*). Ensure that the *Map text labels to pins* field on the *Map* tab has been set correctly. You can also check to ensure that the GDSII layout data has been imported into the abstract generator on correct layers.

Description: A pin has already been created with the same name on the same layer.

ABS-12008

Message: Cell *name*: No value has been specified in the *Map text labels to pins* field on the *Map* tab for the bin *name*. Therefore, the default value *defaultvalue* will be used for mapping text labels to pins.

Description: Unless there are already terminals present, the *Map text labels to pins* specification is required to generate a valid pins view. If you do not specify this, Abstract Generator automatically determines a suitable default value before processing.

ABS-12009

Message: The *Map text labels to pins* field has been left blank and the abstract generator is unable to determine a default text pin mapping for the bin *name* because the cell layout does not contain text labels intersecting any geometry. The abstract generator tries to calculate a default value by examining the layers with text labels and then finding the metal layers that have shapes that intersect any of the text labels. If the abstract generator finds a shape under a text label on a metal layer, it sets the *Map text labels to pins* option to map those text labels to that metal layer. In the current situation, the abstract generator cannot find any metal shapes under the text labels.

ABS-12010

© 2023 Cadence Design Systems, Inc.

Message: Cell *name*: The prBoundary cannot be calculated even though the prBoundary creation has been enabled on the *Boundary* tab. This is because no layers have been specified in the *Using geometry on layers* field on the *Boundary* tab. Specify the appropriate layers for prBoundary creation. See Setting PR Boundary Creation Options in Standalone Abstract Generation.

ABS-12011

Message: Cell *name*: No prBoundary has been created even though the prBoundary creation has been enabled on the *Boundary* tab. This is because no geometry exists on the layers specified in the **U***sing geometry on layers* field on the *Boundary* tab. Specify the appropriate layers for prBoundary creation. See Setting PR Boundary Creation Options in Standalone Abstract Generation.

ABS-12012

Message: Cell *name*: Multiple prBoundaries have been created based on the value specified in the *Using geometry on layers* field on the *Boundary* tab. Update this value so that a single prBoundary is created.

Troubleshooting: Correct the value in the *Using geometry on layers* field to enable Abstract Generator to create a single PR boundary.

ABS-12013

Message: Cell *name*: the calculation of the prBoundary failed even though the prBoundary creation has been enabled on the *Boundary* tab. Ensure that the layers on which the calculation is to be based are specified in the *Using geometry on layers* field on the *Boundary* tab. The specified layers should lead to the creation of a single prBoundary. Additionally, ensure that geometry exists on the specified layers. See Setting PR Boundary Creation Options in Standalone Abstract Generation. See also ABS-12010 and ABS-12011.

ABS-12015

© 2023 Cadence Design Systems, Inc.

Message: Cell *name*: the prBoundary does not enclose all cellview geometries. This is because not all layers with geometry for calculating prBoundary have been specified in the *Using geometry on layers* field on the *Boundary* tab. Ensure that all the required layers have been specified in this field. See Setting PR Boundary Creation Options in Standalone Abstract Generation.

ABS-12016

Message: The *Create boundary* option on the *Boundary* tab has been set to off, but no prBoundary exists in cell. Set the *Create boundary* option to either 'always' or 'as needed'.

Description: The *Create boundary* option should be set to *off* only if a prBoundary already exists in a cell. This message is displayed if no boundary exists and there is no request to create one.

Troubleshooting: For information about creating place-and-route boundaries and specifying the layers to be used in the necessary calculations, see *Boundary* Tab.

ABS-12019

Message: Cell *name*: database could not create pin named *pin name*. Verify the permissions on the design and ensure that write permission exists.

Description: The database is unable to create a pin with the specified name. This is because the write permissions on the design does not exist.

ABS-12020

Message: Cell *name*: database could not create net named *net name*. Verify the permissions on the design and ensure that write permission exists.

Description: The database is unable to create a net with the specified name. This is because the write permissions on the design does not exist.

© 2023 Cadence Design Systems, Inc.

ABS-12022

Message: Cell *name*: the existing terminal *terminalname* was ignored based on the *Exclude existing terminals* field on the *Map* tab. However, no equivalent terminal was created based on the option specified in the *Map text labels to pins* field.

Description: This warning is issued when the terminals specified in the *Exclude existing terminals* field on the *Map* tab are not created back from the equivalent text labels existing in the layout.

ABS-12023

Message: Cell *name*: the pin *name* has shape on an invalid user layer numbered *number* in the technology file. The pin will be deleted from the layout.

Description: This warning is issued during the Pins step when Abstract Generator comes across a pin that exists on an invalid technology layer. Abstract Generator deletes such pins from the layout after issuing this warning.

ABS-12024

Message: Cell *name*: no pin shape will be created on the layer *name* because the layer does not have pin purpose defined in the technology file.

© 2023 Cadence Design Systems, Inc.

Description: This error message is issued when Abstract Generator attempts to add pin shapes on a layer that does not have a purpose defined in the technology file.

Abstract Generator might attempt to create pins in the following ways:

- From the text labels that overlap shapes on the specified layer-purpose pairs. For more information, see Text Labels-to-Pins Mapping.
- From preexisting terminals and pins.
- From the figures and instance pins attached to a terminal's net. This happens when a cellview contains terminals but no pins.

Troubleshooting:

- If you want the shapes to be added on the specified layer, define the missing LPPs in the `techLayerPurposePriorities` section of the technology file by using the Technology File Manager.
- If you want Abstract Generator to ignore the existing terminals without pins, do one of the following:
  - Remove the nets associated to the terminals from the cellview.
  - Specify the terminal names in the *Exclude existing terminals* field on the Map tab.
- Check the value specified for the *Map text labels to pins* field if the error occurs when the pins are being generated through text labels.

ABS-12025

Message: Cell *`name`*: Terminals *`name`* and *`name`* are must join terminals but have been mapped to different terminal types. *`terminalname`* has been specified as *`type`* terminal and *`terminalname`* has been specified as *`type`* terminal. Check the regular expressions specified on the *Map* tab.

ABS-12027

© 2023 Cadence Design Systems, Inc.

Message:	Cell `name`: the existing terminal *name* specified in the *Exclude existing pins on layers* field does not have any pin shape. This terminal will be deleted. The equivalent terminal will be created based on the value specified in the *Map text labels to pins* field on the *Map* tab.
Description:	This warning message is issued when a terminal on the layer specified in the *Exclude existing pins on layers* field does not have a pin.

Related Topics

Specifying Pin Mappings for Abstract Generation

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

4

Extract Step in Standalone Abstract Generation

The Extract step is the second step in the abstract generation process, traces the connectivity between shapes and terminals, starting at the *pin* purpose shapes created in the Pins step. The shapes created during extraction are created on the purpose *net* at the top level of the extract view.

Abstract Generator performs the following functions during the Extract step:

- Extracts each terminal net, one shape at a time.
- Constructs the correct database model for strong, weak, and must-join pins.
- Creates library process antenna information for custom blocks and standard cells.

To run the Extract step:

1. Select one or more cells in the main window and click the *Extract* button. Alternatively, select *Flow – Extract*. The *Flow – Extract* command runs as far the Extract step for the selected cells.
2. When you select the command, the Running Form is displayed. Use this form to view and modify the bin options relevant to the Extract step and any prior steps that must be run.
3. Click *Run* to run the step when you are satisfied with the options settings.

The *Percent Complete* message box reports the progress of the step.

Abstract Generator processes the cells as far as the Extract step, creating an extracted view, along with any of the predecessor views required.

If you have not already generated pins for the selected cells, Abstract Generator automatically runs the Pins step prior to running the Extract step.

You can enable or disable extraction for signal and power nets or specify the extraction to be solely performed for antenna data. The default settings depend on a cell's bin type, and by default, all bins except the *Core* bin have extraction turned off. When the Extract step is

© 2023 Cadence Design Systems, Inc.

complete, Abstract Generator places a graphical icon against the cell to indicate the status of the Extract view that was created. Abstract Generator automatically detects shapes placed in WSP regions and ignores these shapes during net extraction. Therefore, no pins are created for such shapes. You do not need to specify extraction limits for antenna data calculation.

Running Form

The Running form appears whenever you run a flow step. The form is a modified version of the Bin Options form but offers access only to those bins and options relevant to the step or steps about to be run.

The label in the title bar indicates which step is about to be run, such as *Running Step Pins* indicate that the Pins step would be executed. The *Step Selector* provides access only to the options associated with those steps. The *Bin Selector* lists only those bins in which the selected cells reside. See Setting Bin Options in Abstract Generator.

Processing Cells in Different States

When you press a toolbar button to perform one of the main flow steps, Abstract Generator checks the selected cells to ascertain which steps have to be run for each cell.

Suppose you have two cells selected in the *Cell* pane, `Cell1` and `Cell2`. `Cell1` has run as far as the Pins step and Cell2 as far as the Extract step. If you now run the Abstract step for both the cells, the following steps also must be run:

- Extract and Abstract steps for `Cell1`
- Abstract step for `Cell2`

Consequently, the Running form will include all the options related to the Extract and Abstract steps. You can access options of a step that has already been run. If you change an option for a step that has already been run for a particular cell, that step and all downstream steps for that cell will become invalid.

In the example above, you can also access the *Pins* options, even if all the cells have valid pins views. Changing one of the *Pins* options will render the existing pins and extract views for `Cell2` invalid, and these steps will be rerun for that cell.

Related Topics

Specifying Layers and Geometries for Extraction

Exclude Metal Purpose Blockages in Layout Designs

© 2023 Cadence Design Systems, Inc.

Pins Step in Standalone Abstract Generation

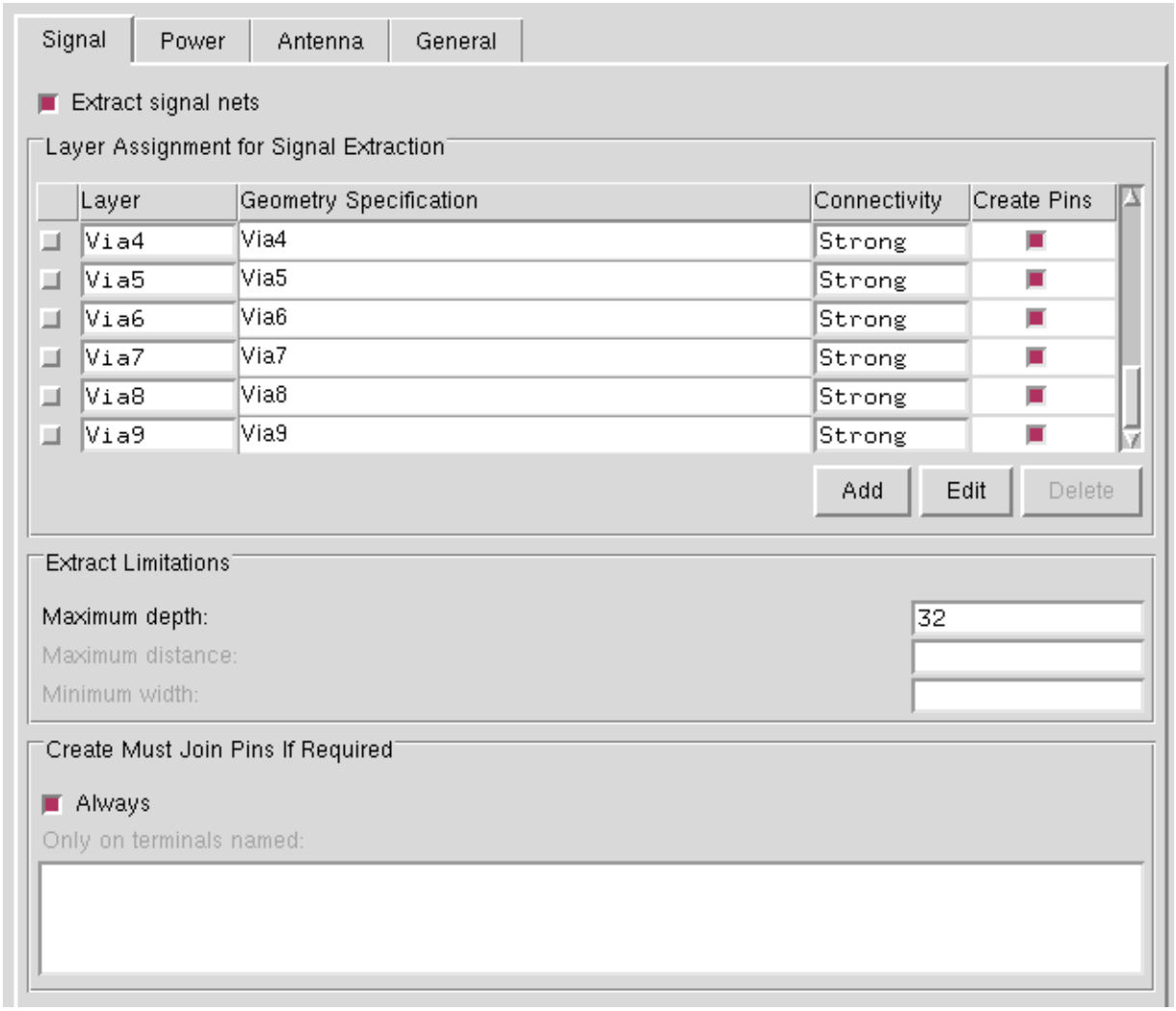
Abstract Step in Standalone Abstract Generation

© 2023 Cadence Design Systems, Inc.

Specifying Layers and Geometries for Extraction

To extract geometries on signal or power nets or to calculate process antenna information, you need to provide information about them to Abstract Generator. The Extract step provides three different tab locations - *Signal*, *Power*, and *Antenna* - for you to provide these details.

1. Choose *Flow – Extract*. The *Signal* tab is the first tab in the Running step Extract form.

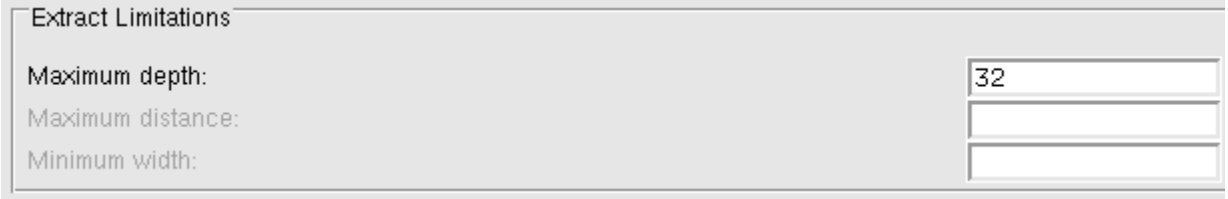


2. Select the *Extract signal nets* option to direct Abstract Generator to run connectivity extraction for signal nets.

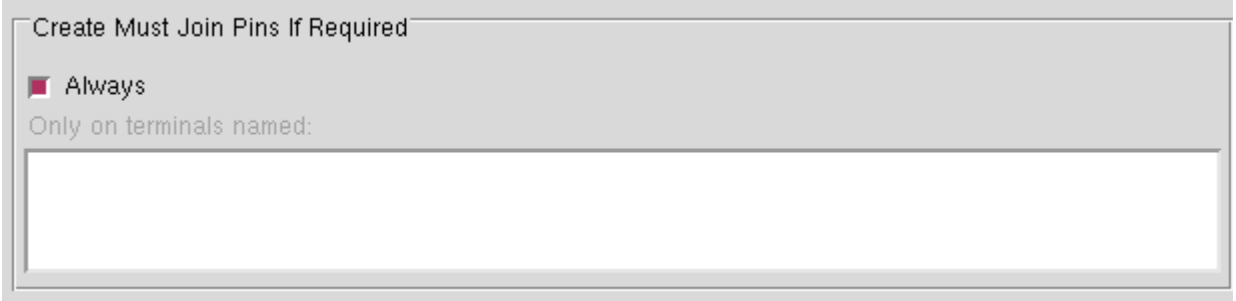
   Selecting the *Extract signal nets* option activates the *Layer Assignment for Signal Extraction* table.

© 2023 Cadence Design Systems, Inc.

3. In the *Layer Assignment for Signal Extraction* table, specify the layers through which you want Abstract Generator to extract. You can also specify the geometry or shapes that should be present on a particular layer when signal net extraction is performed.
4. In the *Layer Assignment for Signal Extraction* table, specify the *Layer* on which signal net extraction is to be performed, and set the *Geometry Specification* for the layers specified in this table.
5. In the *Connectivity* column, you can specify the type of connectivity setting, *Strong* or *Weak*, for a particular layer. The connectivity setting is valid for a layer only if a geometry specification is provided for that layer.
6. Use the *Add*, *Edit*, and *Delete* buttons and the G-Spec Builder Form in conjunction with the Operators to work with the *Layer Assignment for Signal Extraction* table.
7. Use the *Maximum depth*, *Maximum distance*, and *Minimum width* fields for specifying the extraction limits.



8. In the *Create Must Join Pins If Required* section, you can control what happens when the extractor finds two disjoint groups of shapes extracted for the same net.



   The *Extract signal nets* option should be selected in order to generate the must-join pins. If the *Extract signal nets* option is `OFF`, then the *Create Must Join Pins If Required* options do not have any effect.

9. Use the *Always* option and the *Only on terminals named* field to create LEF must-join relationships between disjoint geometries extracted from separate starting pins on the same net.

© 2023 Cadence Design Systems, Inc.

You can face the following issue while using the Running Step Extract form and the following error message might be displayed.

ABS-23005

Cell *name*: Terminal *name* has been ignored although it has the `mustJoin` attribute because it was not indicated in the *Create Must Join Pins If Required* section on the Signal page.

Troubleshooting: To ensure that must-join pins are created for terminals, select the *Always* option or include specific terminals in the *Only on terminals named* field.

Related Topics

Running Step Extract Form

Layer Assignment Tables

Specifying General Options in G-Spec Builder Form

Relationship between Signal and Antenna Tabs

Must-Join and Must-Connect Relationships for Terminals

© 2023 Cadence Design Systems, Inc.

Layer Assignment Tables

The layer assignment tables are accompanied with three buttons - *Add*, *Edit*, and *Delete* to enable you to work with the table. In addition, the G-Spec Builder Form enables you to construct expressions for the geometry specifications.

The entries in the *Layer Assignment for Signal Extraction* table are used by other functions even when the *Extract signal nets* option is switched off. The situations in which this might happen are discussed below.

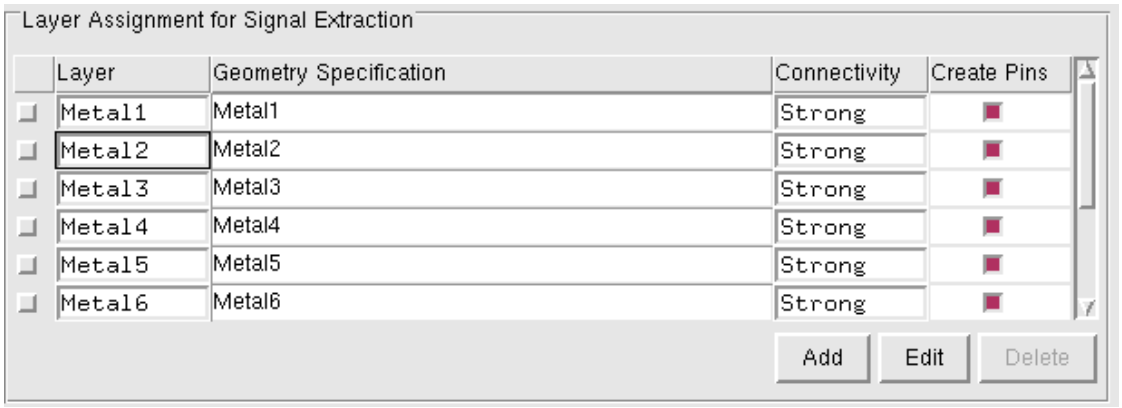
- In the Extract step, if you have a geometry specification with more than two parts, for example as in,

  ```
  (metal1 andnot marker)
  ```

  Abstract Generator finds shapes that touch or overlap the pin shapes. This can result in the creation of pins that are different from the ones created in the Pins step.

  In the above geometry specification example, it might happen that the design has two nets shorted together on `metal1` and the layout has a `marker` layer that indicates the boundary between the two nets. Therefore, any shape on the `metal1` layer and not the `marker` layer which touches an original pin is turned into a pin.

- In the Abstract step, Abstract Generator tries to stretch the pins. To do this, it uses the geometry specification entered in the table to ensure that it converts only the shapes that satisfy the specification into pins.



Ignore Cells at the Top Level during Extraction

Extremely detailed and complicated layout designs might take too much time for extraction. To reduce the extraction time of such layout designs, you can selectively stop the extraction of certain cells at the top-level.

© 2023 Cadence Design Systems, Inc.

Use the `absSetOption` option to specify the cells to be ignored during extraction. Specify the cell names to be ignored as a space separated string in this option. For example, to ignore all instances of `"CELL_55"` and `"CELL_56"` during extraction, use:

```
absSetOption("IgnoreCellsForExtraction" "CELL_55 CELL_56")
```

The default state of this option is:

```
absSetOption("IgnoreCellsForExtraction" "")
```

For ignoring fluid guard rings and Multi-Part Paths (MPPs) that are implemented as Pcell instances, you can set the corresponding cell names in this option. Consequently, all instances of the specified cells are ignored during extraction.

Exclude Metal Purpose Blockages in Layout Designs

A layout design containing shapes on a metal purpose blockage might cause problems during extraction unless you modify the geometry specification settings. This is because these shapes usually cover large areas of a design so that when extraction is performed through these layers, all the geometry under the blockage is also extracted. This would obviously impact extraction performance and could cause shorts.

To avoid this potential problem, you can specify the purposes to be ignored in the *Ignored Purpose(s)* field (*File – General Options – General* tab). By default, shapes on the boundary purpose are ignored. If you want to include the shapes on the ignored purposes during the Pin, Extract, and Abstract steps, you can explicitly mention the layer and the purpose names in the geometry specification at the appropriate locations in the flow steps.

Enabling short detection during signal or power extraction can have a significant performance impact on the extraction process and the abstract generation runtime. To avoid this, you can set the `ExtractDisableShortDetection` option to `true`, which means that short detection is not run as a part of the signal or power extraction process.

Extract Signal Nets for Blocks and IO Cells

It is recommended that you should not use the *Extract signal nets* option for blocks or IO cells unless you are sure you need to. For example, use this option only if you want to perform antenna extraction.

If you do need to extract signal nets, do not create pins on a given layer unless it is necessary. For example, do not create pins on polysilicon, diffusion, or contact layers, except where there are dominant cut-layer spacings.

© 2023 Cadence Design Systems, Inc.

You should also turn on the *Create boundary pins* option in the Abstract step to ensure that not all of the extracted geometry is turned into pins. If you do not, Abstract Generator would cut around each pin, which would increase the time taken to generate the blockage model.

Related Topics

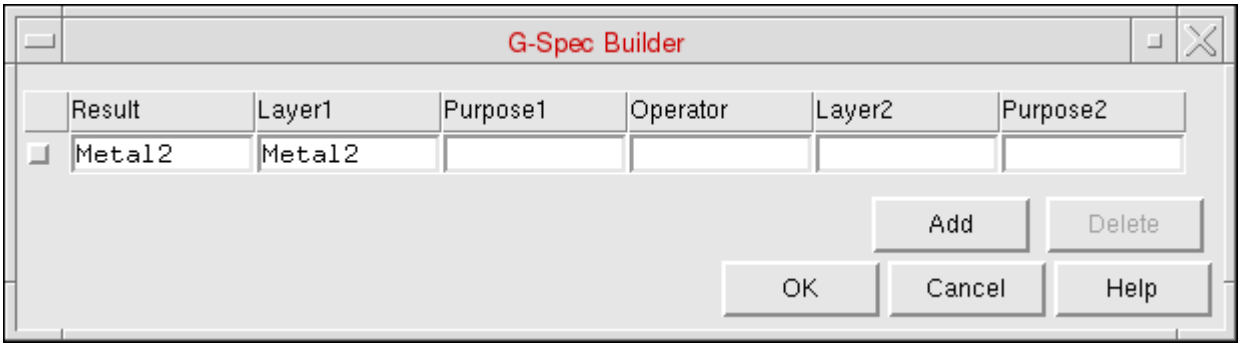
Customizing Pin Shapes in Standalone Abstract Generator

Relationship between Signal and Antenna Tabs

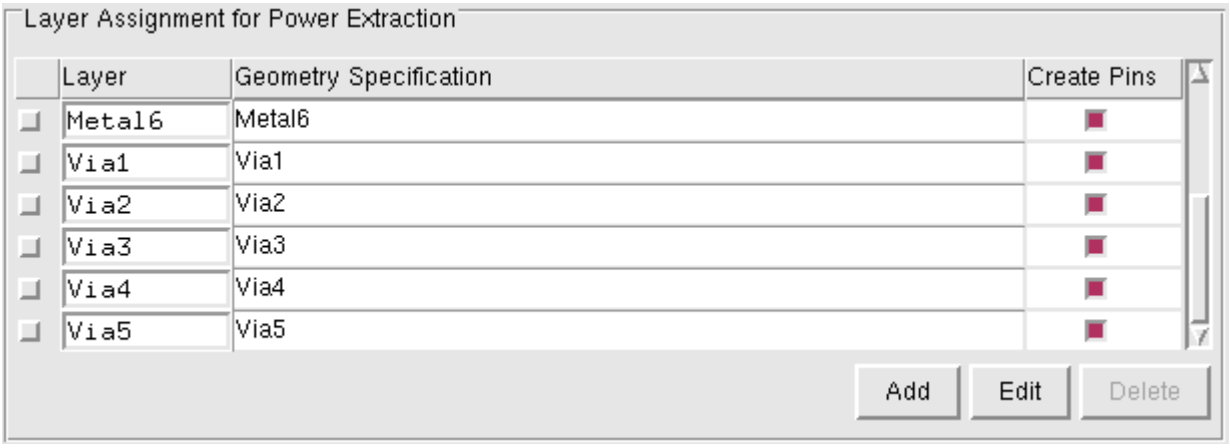
© 2023 Cadence Design Systems, Inc.

Specifying General Options in G-Spec Builder Form

The G-Spec Builder form allows less experienced users to build their own geometry specifications (g-specs) using temporary layers.



1. Click *Add* to add a new blank row.
2. Click *Delete* to remove any rows selected for deletion. Selecting the check box at the far left of each row marks a row for deletion.
3. You can create a g-spec only for one layer at a time. If you want to create g-specs for multiple layers, you must enter the first g-spec you have specified into the original table by clicking *OK*.
4. Click *Geometry Specification* on the next layer you want to create a g-spec for.



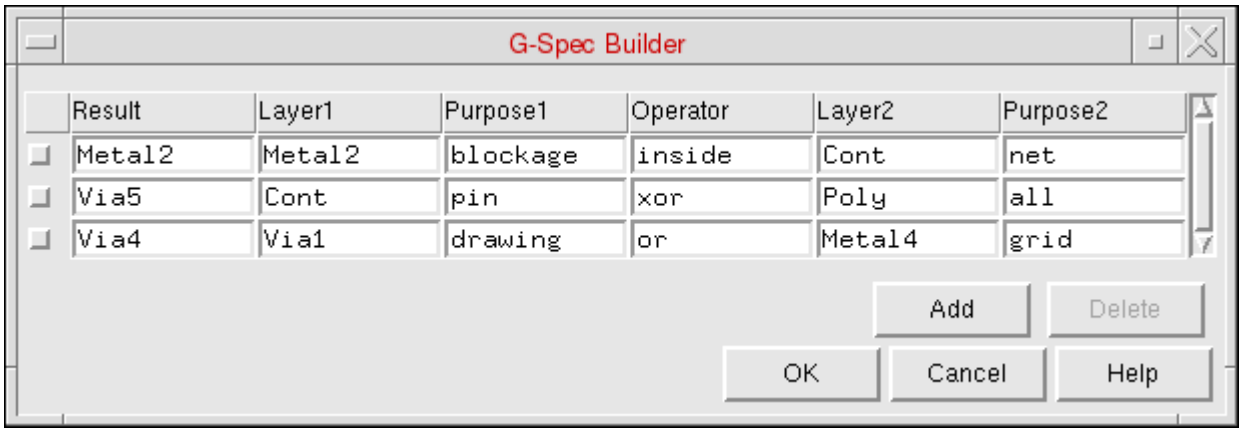
5. Click *Edit* again to reopen the G-Spec Builder form.
6. Click *OK*.

These temporary layers are translated into the correct g-spec syntax.

© 2023 Cadence Design Systems, Inc.

For g-specs that are more complex than two layer-purpose pairs and an operator, it is necessary to use temporary layers. To do this, you must first define a temporary layer to be the equivalent of a g-spec. Its definition can then be used at any future time within the G-Spec Builder form. When it is translated into proper syntax, the temporary layer definition is substituted.

For example, consider the following entries in the G-Spec Builder form.



The above g-spec would translate into the *Layer* and *Geometry Specification* fields in the main layer assignment table as described below.

Layer	Geometry Specification
METAL1	((METAL1 pin or METAL1 net) size 0.5) and METAL1 drawing)

Take care while creating g-specs to act on pins on the *prBoundary* layer. This layer can contain many thousands of shapes and an incorrect g-spec can cause them to merge into a single rectangle for each pin.

Operators

The following is a list of the Boolean operators that can be used in the G-Spec Builder form.

```
geomSpec    =   layer | layer-purpose | geomSpec binaryOp
                geomSpec | geomSpec size float | geomSpec
                holes | noholes | (geomSpec)
```


© 2023 Cadence Design Systems, Inc.

```
binaryOp    =   andnot | and | or | xor | inside | outside |
                straddle | size
```

Examples

```
M1
(M1 or M1 pin)
(M1 size 0.5)
(M1 noholes)
```

The operators are described in detail below.

Operator	Description
`holes`	A hole is an area created when the perimeter of a polygon touches itself, enclosing an area that is not the polygon. These are sometimes referred to as donut holes. The `holes` operator generates shapes consisting of all the holes in shapes of the input layer.
`noholes`	The `noholes` operator generates new shapes consisting of all the original shapes of the input layer without the holes.
`andnot`	The `andnot` operator generates new shapes from areas of the first input layer that do not overlap any area of the second input layer.
`and`	The `and` operator generates new shapes from the overlap of original shapes on the input layers.
`or`	The `or` operator merges all shape layers. For polygon layers, a merge generates polygons whose area encompasses all the areas from all the layers and removes all the edges internal to the polygons.
`xor`	The `xor` operator generates new shapes from those portions of either input layer that do not overlap the other layer.
`inside`	The `inside` operator selects the shapes on the first layer that are totally inside shapes of the second layer. A first-layer shape is inside if its entire area is covered by the area of a second-layer shape. Coincident shapes are considered inside in this check. If shapes on two layers are coincident on all sides, each layer can be defined as being inside the other.
`outside`	The `outside` operator selects the shapes on the first layer that are totally outside shapes of the second layer. A first layer shape is outside if none of its area is covered by the area of a second layer shape. Abutting edges are considered outside.

© 2023 Cadence Design Systems, Inc.

Operator	Description
`straddle`	The `straddle` operator selects shapes on the first layer that straddle shapes on the second layer. A first-layer shape is straddling if part of its area is covered by a second layer and part is not. Abutting and coincident edges do not affect this function.
`size`	The `size` operator “grows” the shapes on the input layer by the specified amount. If the size amount is negative, the shapes will be shrunk by the specified amount. The resultant shapes are automatically merged.

Related Topics

G-Spec Builder Form

Specifying Layers and Geometries for Extraction

© 2023 Cadence Design Systems, Inc.

Relationship between Signal and Antenna Tabs

It is important to understand how options on different tabs—*Signal* and *Antenna* are interrelated and impact the overall extraction.

The *Antenna* tab in the Running step Extract form provides various calculation options for calculating the input, output, and tristate antenna pins and the antenna metal and side areas. If any one of the calculation options is selected, full extraction is performed for establishing antenna properties.

You must specify the polysilicon, contact cut, and diffusion layers, along with a suitable expression for the diffusion layer, such as `(diffn (diffn andnot poly))`, in the Layer Assignment Tables. If you do not provide this information, the extraction process will not be able to locate any polysilicon or diffusion geometry that is required to perform the subsequent antenna calculation and gate/drain recognition. It is not necessary to provide a complex expression for the polysilicon layer; it has to be listed as an extract layer. You should ensure that diffusion, contact, or polysilicon layer information is not entered in the pin layer field on the *Map* tab because you only want to extract through these layers and not create pins on them.

It is feasible to turn off the *Extract signal nets* option at this point. This would result in Abstract Generator performing a full extraction in the background to calculate the antenna properties specified in the *Antenna* tab. The result would be that the pins created would be the same as those created after the Pins step, that is, no new pins would be created. Effectively, in this scenario, a “behind-the-scenes” full extraction is being performed solely for the purpose of calculating antenna numbers. Alternatively, if the *Extract signal nets* option is turned on, full extraction will be performed and the geometry found during the extraction will be turned into pin geometry.

The table below shows the type of extraction that Abstract Generator would perform based on the options selected on the *Signal* and *Antenna* tabs in the Extract step.

"Extract signal nets" option on the Signal Tab	**Any "Calculate" option on the Antenna Tab**	**"Use different layer assignments for antenna calculations only" option on the Antenna Tab**	**Type of Extraction Performed**
Off	Off	Off	Preset pin extraction

© 2023 Cadence Design Systems, Inc.

"Extract signal nets" option on the Signal Tab	Any "Calculate" option on the Antenna Tab	"Use different layer assignments for antenna calculations only" option on the Antenna Tab	Type of Extraction Performed
On	Off	Off	Signal extraction only using the values specified in the *Layer Assignment for Signal Extraction* table.
Off	On	Off	Antenna extraction only using the values specified in the *Layer Assignment for Signal Extraction* table.
Off	Off	On	No extraction performed
On	On	Off	Antenna and signal net extraction using the values specified in the *Layer Assignment for Signal Extraction* table.
Off	On	On	Antenna extraction only using the values specified in the *Layer Assignment for Signal Extraction* table.

© 2023 Cadence Design Systems, Inc.

"Extract signal nets" option on the Signal Tab	Any "Calculate" option on the Antenna Tab	"Use different layer assignments for antenna calculations only" option on the Antenna Tab	Type of Extraction Performed
On	On	On	■ Signal and Antenna Layers are the Same The only extraction performed in this case is that using the values specified in the *Layer Assignment for Signal Extraction* table to calculate both antenna and signal nets extraction. ■ Signal and Antenna Layers are Different Two extractions are performed in this case. Antenna extraction is performed first using the values specified in the *Layer Assignment for Antenna Extraction* table, followed by signal net extraction using the values specified in the *Layer Assignment for Signal Extraction* table.

The only other situation where two types of extractions are possible is when the *Maximum depth* option is set to a value different from the default value *20*. In this situation, the rules mentioned in the table above would apply.

Related Topics

Layer Assignment Tables

© 2023 Cadence Design Systems, Inc.

Must-Join and Must-Connect Relationships for Terminals

You can set the following options in the `abstract.options` file to specify must-join and must-connect relationships for terminals:

`ExtractMustJoinAlways` or `ExtractMustJoinTerminals`

The default value of `ExtractMustJoinAlways` is `false`. When set to `true`, if Abstract Generator finds disjoint groups of shapes extracted for any nets in the layout, it renames the terminals, and sets the `mustJoin` attribute on these terminals. It corresponds to the *Always* checkbox in the Extract page.

Alternatively, list specific terminals with the `ExtractMustJoinTerminals` option. By default, the list is empty. This option corresponds to the *Only on terminals named* field in the Extract page.

`ExtractMustConnectAllPinsAlways` or `ExtractMustConnectTerminals`

The default value of `ExtractMustConnectAlways` is `false`. When set to `true`, if Abstract Generator finds two disjoint groups of shapes extracted for any net in the layout, it sets the same terminal name for all the pins under it, and applies the `mustConnectAllPins` attribute on these terminals.

Alternatively, list specific terminals with the `ExtractMustConnectTerminals` option. By default, the list is empty.

These options do not have corresponding fields in the Abstract Generator graphical user interface.

Must-connect options take precedence over the must-join options.

Related Topics

Specifying Layers and Geometries for Extraction

© 2023 Cadence Design Systems, Inc.

Antenna Generation for MUSTJOIN and MUSTJOINALLPORTS Terminals

Abstract Generator computes the antenna data for a given layer. The antenna parameters for some of these layers are defined in the LEF technology file. The main antenna calculation is performed at the end of the Extract step, where Abstract Generator computes antenna data.

During antenna calculation when extraction is performed using the Extract step, a single island of overlapping or connected shapes is created for each terminal. In this island, all shapes are connected as per the layer connectivity model. Antenna numbers are calculated according to shapes in the island, which depends on their connectivity to other shapes.

If some preexisting constraints are preventing all the shapes to be connected, you can use either the `MUSTJOIN` setting or the `MUSTJOINALLPORTS` property to connect all the shapes from outside. These settings let the router connect all the shapes from outside when no space is available for the metal shapes inside the cellview.

Related Topics

Calculation of Antenna Numbers for MUSTJOIN Terminals

Calculation of Antenna Numbers for MUSTJOINALLPORTS Terminals

Calculation of Antenna Numbers for MUSTJOIN Terminals

To calculate the antenna numbers of MUSTJOIN terminals, in the Extract step, Abstract Generator identifies disjoint groups of shapes extracted for nets in the layout. It then renames these terminals and sets the `MUSTJOIN` attribute on these terminals.

Abstract Generator creates a separate terminal for each island. For example, for terminal A and n number of disjoint islands, the tool creates terminals A, A_mj1, A_mj2 …… A_mjn-1.

For each island, Abstract Generator sets the antenna number to its corresponding terminal name. Abstract Generator calculates the antenna values by comparing different antenna numbers for each island individually. It does not compare antenna numbers across different islands.

You can use the following options to calculate antenna numbers for disjoint islands of a `MUSTJOIN` terminal:

- Either set the `ExtractMustJoinTerminals` option to true for the all the terminals or specify the terminal name in the `ExtractMustJoinTerminals` option.

```
absSetBinOption( "<Bin Name>" "ExtractMustJoinAlways" "true") OR
absSetBinOption( "<Bin Name>" "ExtractMustJoinTerminals" "terminal_name")
```


© 2023 Cadence Design Systems, Inc.

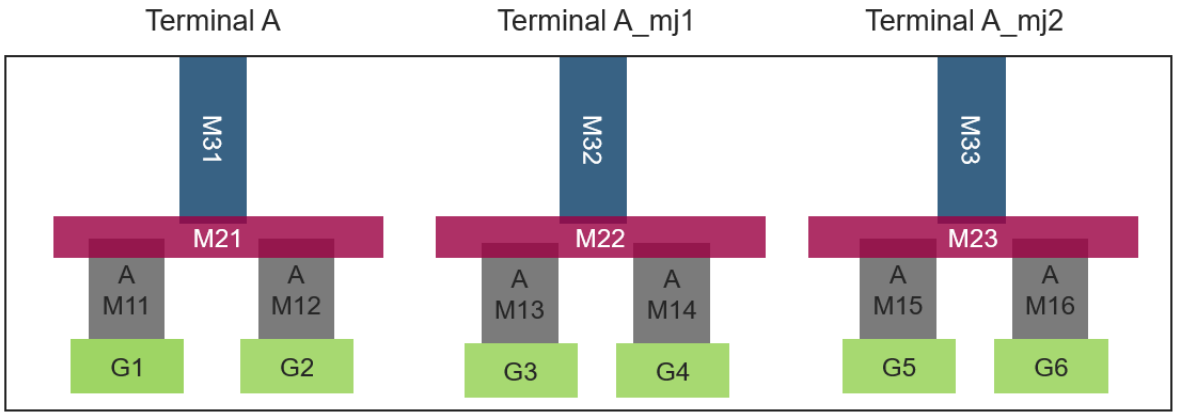
- Select the *Extract signal nets* option on the Signal Tab in the Running step Extract form or use `absSetBinOption` to set the `ExtractSig` option to `true`.

  ```
  absSetBinOption( "<Bin Name>" "ExtractSig" "true")
  ```

- Select one of the following antenna calculation options.
  - *Calculate hierarchical antenna*
  - *Calculate input pin antenna*
  - *Calculate output pin antenna*
  - *Calculate inout pin antenna*
  - *Calculate antenna metal area*
  - *Calculate antenna metal side area*

To understand this better, consider the following example, `Mxy` is `y`th shape of `Mx` layer. For example, `M12` is the second shape of the `M1` layer.

Here, `G1`, `G2`, `G3`, `G4`, `G5`, and `G6` represent the gate areas for the corresponding metal shapes. Cut or via layers are omitted for the sake of simplicity in the design.



When you select the *Calculate antenna metal area* option, Abstract Generator calculates the following values:

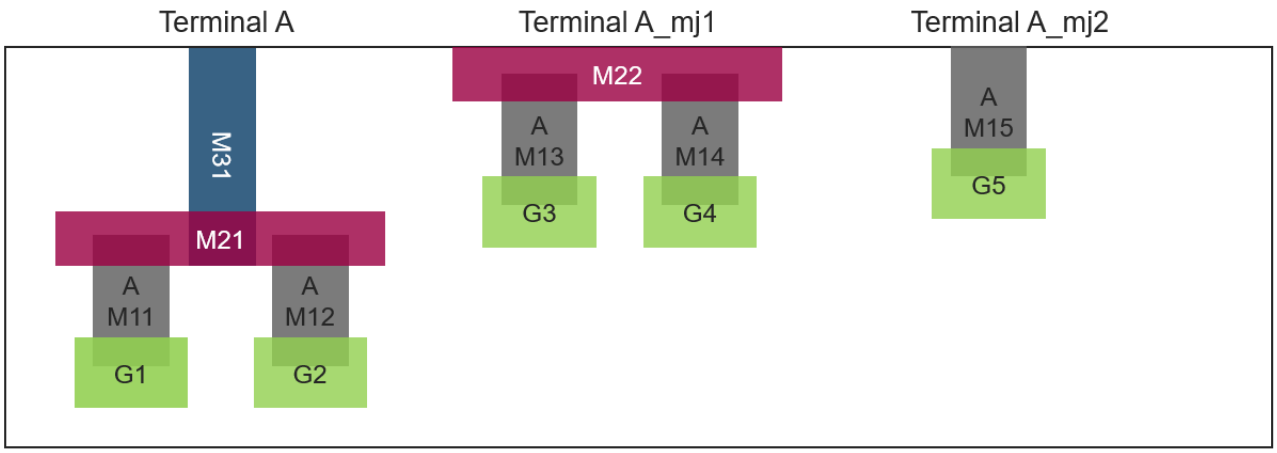
- `ANTENNAPARTIALMETALAREA` is the largest metal shape area for each layer.
- `ANTENNAGATEAREA` is the minimum gate area among all the gates that are visible through each metal shape.
- `ANTENNAMAXAREACAR` is the largest path among all the paths. It is calculated as area of metal layer divided by the area of gates visible to the shape.

© 2023 Cadence Design Systems, Inc.

The following table shows how antenna values are calculated for the above example:

Terminal	ANTENNAPARTIALMETALAREA	ANTENNAGATEAREA	ANTENNAMAXAREACAR
A	M1 : max(M11, M12) M2 : M21 M3 : M31	M1 : min(G1,G2) M2 : G1+G2 M3 : G1+G2	M1 : max(M11/G1,M12/G2) M2 : M21/(G1+G2) + CAR M1 M3 : M31/(G1+G2) + CAR M2
A_mj1	M1 : max(M13, M14) M2 : M22 M3 : M32	M1 : min(G3,G4) M2 : G3+G4 M3 : G3+G4	M1 : max(M13/G3,M14/G4) M2 : M22/(G3+G4) + CAR M1 M3 : M32/(G3+G4) + CAR M2
A_mj2	M1 : max(M15, M16) M2 : M23 M3 : M33	M1 : min(G5,G6) M2 : G5+G6 M3 : G5+G6	M1 : max(M15/G5,M16/G6) M2 : M23/(G5+G6) + CAR M1 M3 : M33/(G5+G6) + CAR M2

Consider another example, as shown below:



The following table shows how antenna values are calculated for the above example:

Terminal	ANTENNAPARTIALMETALAREA	ANTENNAGATEAREA	ANTENNAMAXAREACAR
A	M1 : max(M11, M12) M2 : M21 M3 : M31	M1 : min(G1,G2) M2 : G1+G2 M3 : G1+G2	M1 : max(M11/G1,M12/G2) M2 : M21/(G1+G2) + CAR M1 M3 : M31/(G1+G2) + CAR M2
A_mj1	M1 : max(M13, M14) M2 : M22	M1 : min(G3,G4) M2 : G3+G4	M1 : max(M13/G3,M14/G4) M2 : M22/(G3+G4) + CAR M1
A_mj2	M1 : M15	M1 : G5	M1 : M15/G5

© 2023 Cadence Design Systems, Inc.

Related Topics

Antenna Generation for MUSTJOIN and MUSTJOINALLPORTS Terminals

Calculation of Antenna Numbers for MUSTJOINALLPORTS Terminals

Calculation of Antenna Numbers for MUSTJOINALLPORTS Terminals

To calculate the antenna numbers of `MUSTJOINALLPORTS` terminals, in the Extract step, Abstract Generator extracts multiple disjoint islands of connected shapes for a single terminal. Antenna calculation is done after combining all the islands into a single island using a metal layer.

Abstract Generator creates a single terminal and puts all the shapes from all the islands under the terminal. A single placeholder terminal is created for antenna values of all the disjoint islands.

The router uses the metal layers to connect all the disjoint islands of the terminal from outside the `MACRO` block. If among all the disjoint islands of the terminal `Mtop` is the top-most metal layer, then router uses `Mtop+1` metal layer to connect all the islands by dropping corresponding vias. If `Mtop` is the top-most metal layer in the technology, then router uses `Mtop` metal layer to connect all the islands.

As the routing operation is performed after abstract generation, Abstract Generator assumes that router will use pseudo or dummy metal shapes with zero area to connect all the islands.

For pseudo metal shapes and corresponding layers, Abstract Generator calculates the following:

- `ANTENNAGATEAREA`
- `ANTENNADIFFAREA`
- `ANTENNADIFFWIDTH`
- `ANTENNACREDITS`
- `ANTENNAMAXAREACAR`

During calculation of `ANTENNAMAXAREACAR`, Abstract Generator excludes the area of the pseudo shapes.

You can use the following options to calculate antenna numbers for disjoint islands of a `MUSTJOINALLPORTS` terminal:

- For `MUSTJOINALLPORTS` terminals, do one of the following:

© 2023 Cadence Design Systems, Inc.

- Set the `ExtractMustConnectAllPins` option to `true` for the all the terminals
- Specify the terminal name in the `ExtractMustConnectAllPinsTerminals` option.

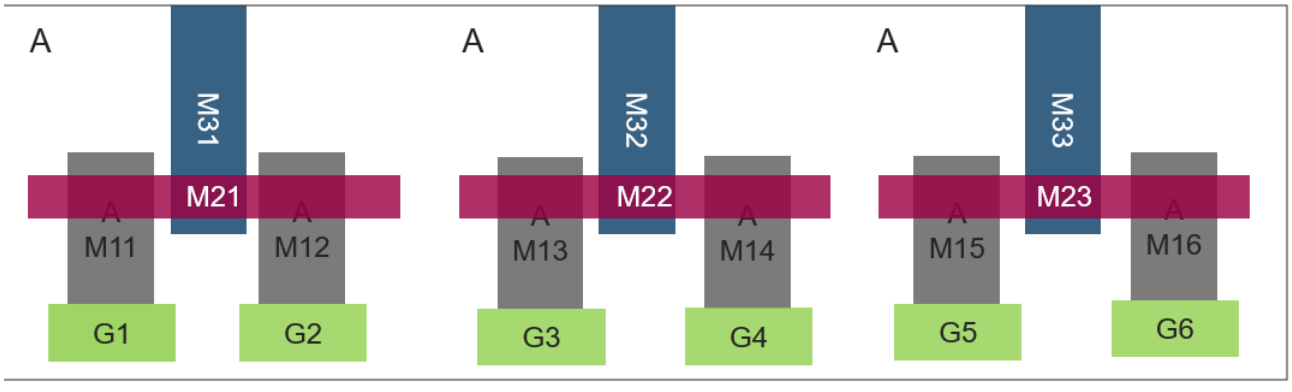
```
absSetBinOption( "<Bin Name>"
"ExtractMustConnectAllPinsAlways" "true") OR

absSetBinOption( "<Bin Name>"
"ExtractMustConnectAllPinsTerminals" "terminal_name")
```

- Select the *Extract signal nets* option on the Signal Tab in the Running step Extract form or use `absSetBinOption` to set the `ExtractSig` option to `true`.

```
absSetBinOption( "<Bin Name>" "ExtractSig" "true")
```

- Select one of the following antenna calculation options.
  - *Calculate hierarchical antenna*
  - *Calculate input pin antenna*
  - *Calculate output pin antenna*
  - *Calculate inout pin antenna*
  - *Calculate antenna metal area*
  - *Calculate antenna metal side area*

To establish a better understanding of how this functionality operates, consider the following example, where terminal A contains three disjoint islands of metal shapes. During antenna calculation, three disjoint set of islands will generate three sets of antenna numbers but OpenAccess terminology allows only a single placeholder terminal that contains all the islands to calculate the antenna area.



`M11` represents the first metal shape of `M1` layer.

© 2023 Cadence Design Systems, Inc.

`M12` represents the second metal shape of `M1` layer.

`M21` represents the first metal shape of `M2` layer.

`M22` represents the second metal shape of `M2` layer.

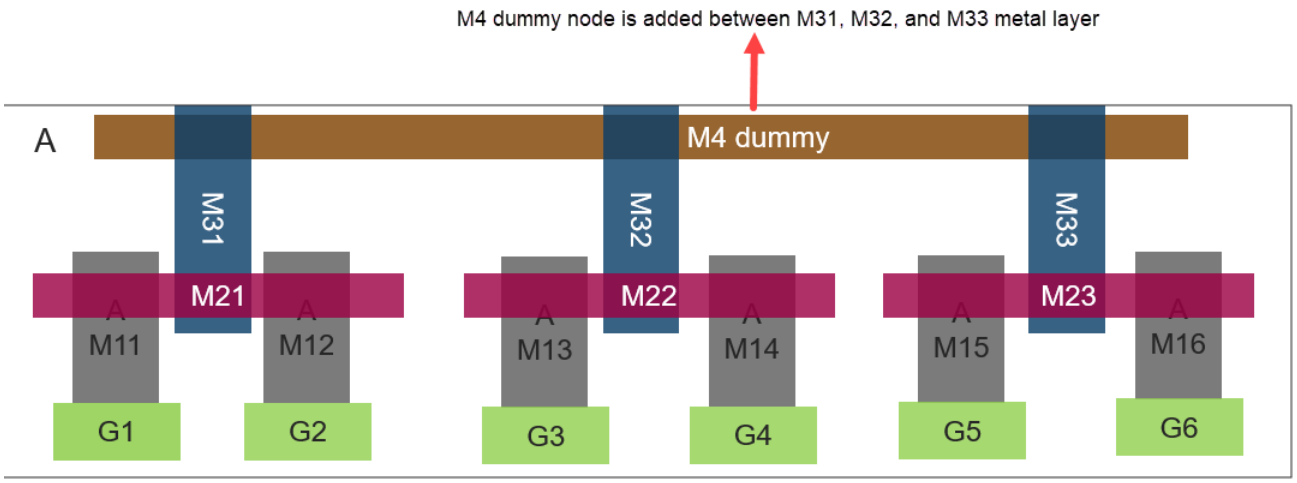
`M31` represents the first metal shape of `M3` layer.

`M32` represents the second metal shape of `M3` layer.

`G1`, `G2`, `G3`, `G4`, `G5`, and `G6` represents the gate areas for corresponding metal shapes.

Note: Cut or via layers are omitted for the sake of simplicity in the design.

Abstract Generator adds dummy nodes at the `Mtop` metal layer with zero area to connect all the islands. Here, the highest metal layer is `M3`, and the tool adds `M4` dummy layer to connect all the islands. The following image represents the combined islands.



When you select the *Calculate antenna metal area* option, Abstract Generator calculates `ANTENNAPARTIALMETALAREA`, `ANTENNAGATEAREA`, and `ANTENNAMAXAREACAR`.

- `ANTENNAPARTIALMETALAREA` represents the largest metal shape area for each layer.
- `ANTENNAGATEAREA` represents the minimum gate area among all the gates that are visible through each metal shape.
- `ANTENNAMAXAREACAR` represents the largest path among all the paths. It is calculated as area of metal layer divided by the area of gates visible to the shape.

© 2023 Cadence Design Systems, Inc.

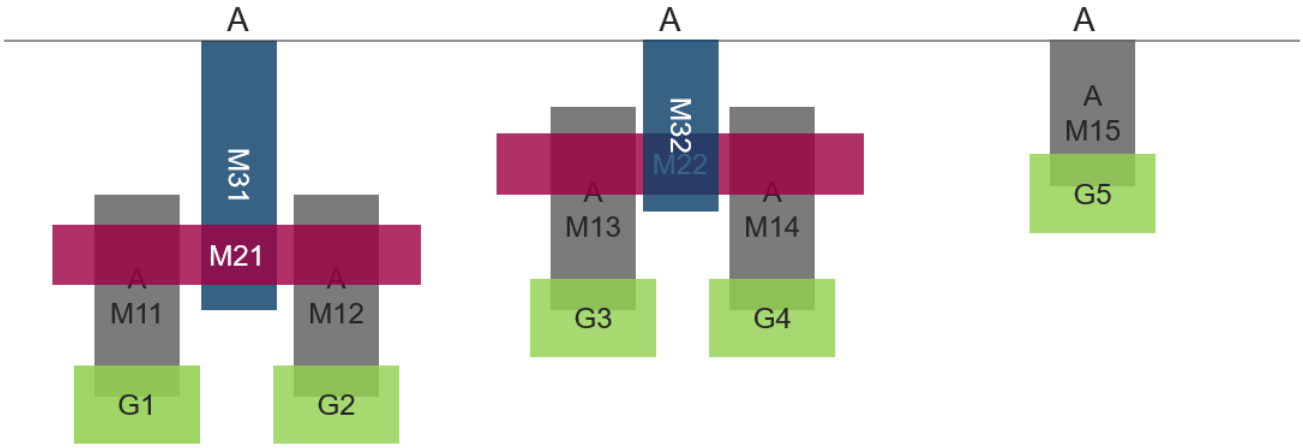
The antenna calculation value is calculated for all the layers above the lowest metal layer on which the pin is present.

ANTENNAPARTIALMETALAREA	ANTENNAGATEAREA	ANTENNAMAXAREACAR
M1:max(M11,M12,M13,M14,M15,M16) M2:max(M21,M22,M23) M3:max(M31,M32,M33) M4:0	M1:min(G1,G2,G3,G4,G5,G6) M2:min(G1+G2,G3+G4,G5+G6) M3:min(G1+G2, G3+G4,G5+G6) M4:G1+G2+G3+G4+G5+G6	M1:max(M11/G1,M12/G2,M13/G3,M14/G4,M15/G5,M16/G6) M2:max(M21/(G1+G2) + PAR M1, M22/(G3+G4) + PAR M1, M23/(G5+G6)) + PAR M1 M3:max(M31/(G1+G2) + M21/(G1+G2) + max(M11/G1+ M12/G2), M32/(G3+G4) + M22/(G3+G4) + max(M13/G3+ M14/G4), M33/(G5+G6)+ M23/(G5+G6)) + max(M15/G5+ M16/G6)) M4:CAR of M3

Consider another example in which terminal A contains three asymmetric islands. The first two islands contain metal layers combined with all the nets but the third island contains only one single metal layer.

© 2023 Cadence Design Systems, Inc.

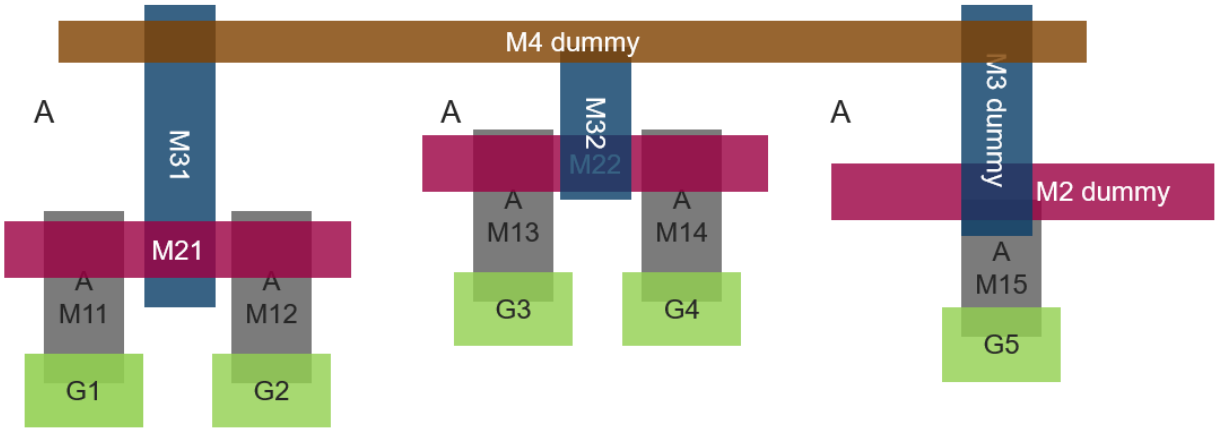
Abstract Generator needs to connect all the islands but as per the connectivity, but `M15` cannot be directly connected to `M32` or `M31`, but `M15` can be connected to `Metal2` and `Metal3` through dummy shapes.



Abstract Generator makes all the islands appear identical by adding `M2` and `M3` dummy shapes to connect the third island. Once the topmost layer of all islands is same and all the islands become identical, the tool adds a new dummy shape `Mtop+1` metal layer, that is `M4` to connect all the disjoint islands and performs antenna calculation accordingly.

When the same top-most layer is added to all the islands, the islands become identical and the tool adds a new dummy shape `Mtop+1` metal layer, `M4` to connect all the disjoint islands and performs the antenna calculation accordingly.

The following image represents combined islands.



© 2023 Cadence Design Systems, Inc.

When you select the *Calculate antenna metal area* option, Abstract Generator calculates ANTENNAPARTIALMETALAREA, ANTENNAGATEAREA, and ANTENNAMAXAREACAR. This value is calculated for all the layers above the lowest metal layer on which the pin is present.

ANTENNAPARTIALMETALAREA	ANTENNAGATEAREA	ANTENNAMAXAREACAR
M1:max(M11,M12,M13,M14,M15) M2: max(M21,M22) M3: max(M31,M32) M4: NA	M1:min(G1,G2,G3,G4,G5) M2: min(G1+G2,G3+G4) M3: min(G1+G2, G3+G4, G5) M4: G1+G2+G3+G4+G5	M1: max(M11/G1,M12/G2,M13/G3,M14/G4,M15/G5) M2: max(M21/(G1+G2) + PAR M1,M22/G3+G4 + PAR M1) M3: max(M31/(G1+G2) + M21/(G1+G2) + max(M11/G1+ M12/G2) M32/(G3+G4) + M22/(G3+G4) + max(M13/G3+ M14/G4)) M4: CAR of M3

Related Topics

Antenna Generation for MUSTJOIN and MUSTJOINALLPORTS Terminals

Calculation of Antenna Numbers for MUSTJOIN Terminals

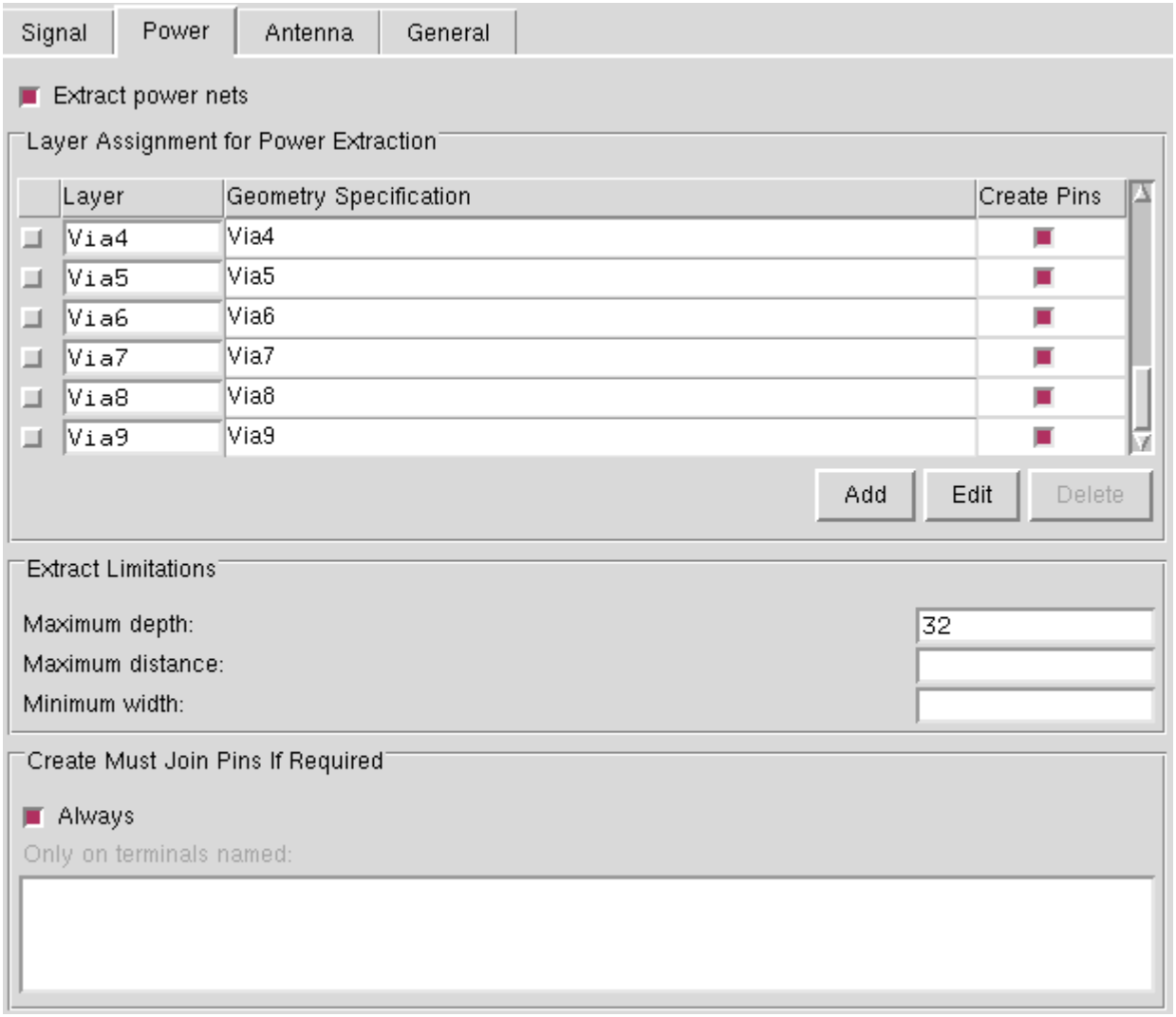
Antenna Calculation by Abstract Generator

© 2023 Cadence Design Systems, Inc.

Extracting Power Nets in Standalone Abstract Generator

To access the *Power* tab in the Running Step Extract form:

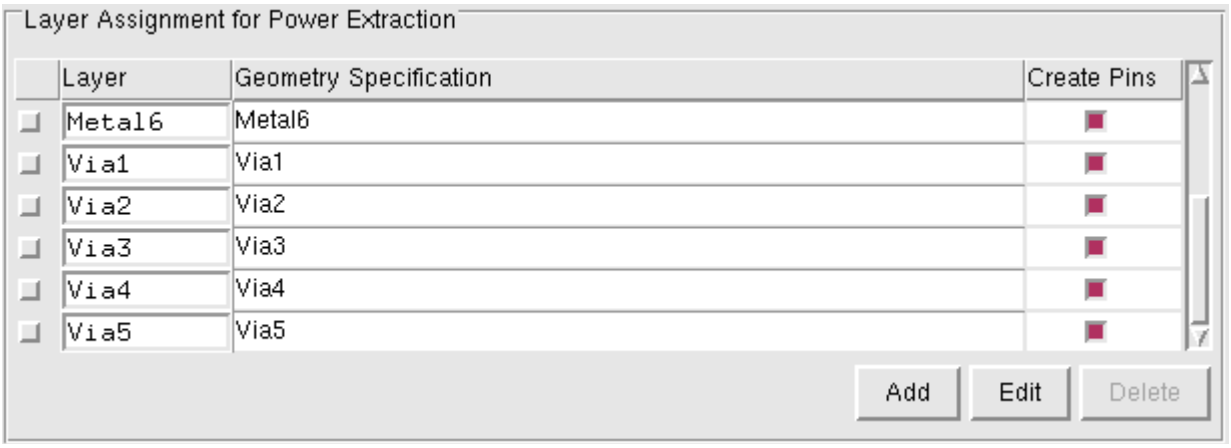
1. Choose *Flow – Extract* and click the *Power* tab.



2. Select the *Extract power nets* option to run connectivity extraction for power nets.

© 2023 Cadence Design Systems, Inc.

3. Use the *Layer Assignment for Power Extraction* table to specify the geometry or shapes that should be present on a particular layer when power net extraction is performed.



   Use this table to specify the Layer on which power net extraction is to be performed. This table becomes active when you select the *Extract power nets* option.

4. Set the *Geometry Specification* for the layers specified in the above table.

5. Select *Create Pins* to create pins corresponding to the layers specified in the *Layers* column.

   Power net extraction for large blocks can often be slow if pins are created on via layers. This is partly due to the large number of via pins that can be created. If power net extraction takes too long, abort the run, re-extract and do not create pins on via layers.

6. Use the *Add*, *Edit*, and *Delete* buttons and the G-Spec Builder Form in conjunction with the operators to work with the *Layer Assignment for Power Extraction* table.

7. Use the *Maximum depth*, *Maximum distance*, and *Minimum width* fields for specifying the extraction limits.

8. The options in the *Create Must Join Pins If Required* section are applicable only to signal nets.

© 2023 Cadence Design Systems, Inc.

Related Topics

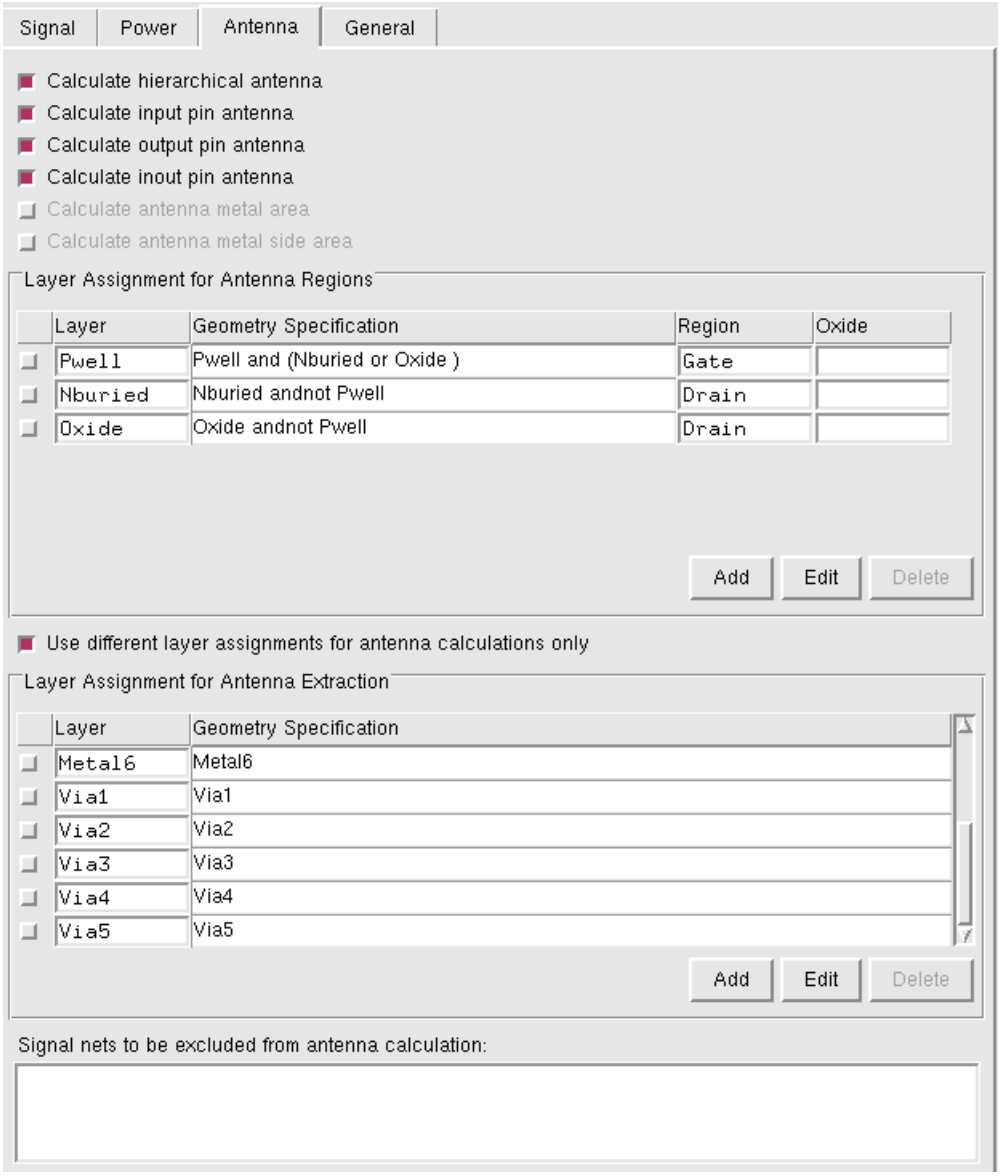
Running Step Extract Form

© 2023 Cadence Design Systems, Inc.

Processing Antenna Information in Standalone Abstract Generator

To access the *Antenna* tab in the Running step Extract form:

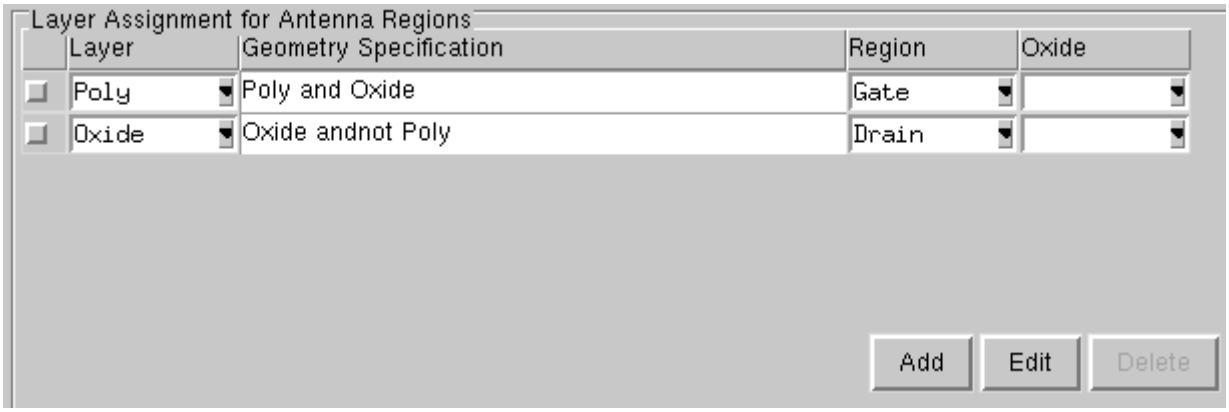
1. Choose *Flow – Extract* and click the *Antenna* tab in the Running step Extract form.



The options on this tab let you create process antenna information for custom blocks and standard cells. By default, all the options on this tab are off.

© 2023 Cadence Design Systems, Inc.

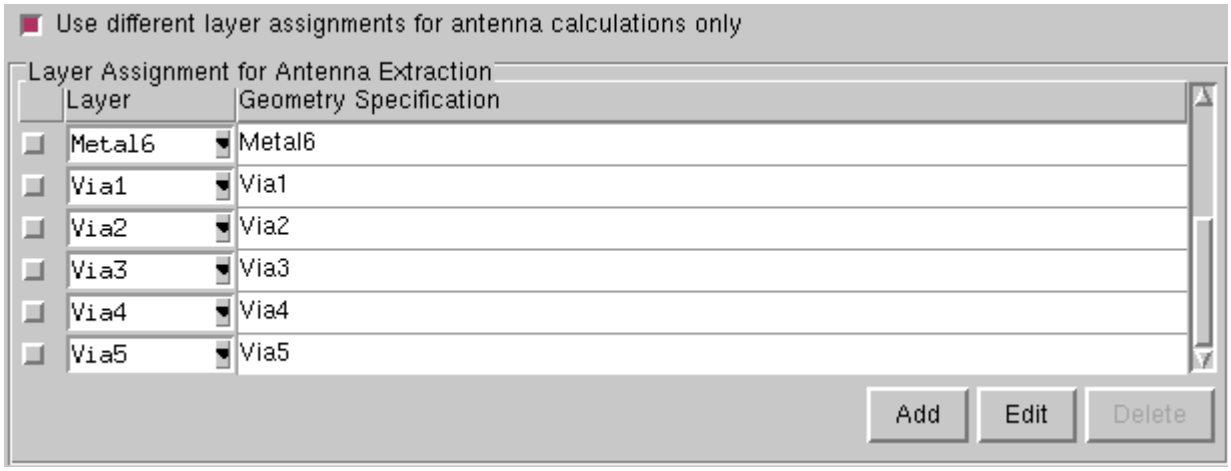
2. Choose one of the following options on the *Antenna* tab for calculating process antenna data:
   - *Calculate hierarchical antenna*
   - *Calculate input pin antenna*
   - *Calculate output pin antenna*
   - *Calculate inout pin antenna*
   - *Calculate antenna metal area*
   - *Calculate antenna metal side area*
3. Select *Layer Assignment for Antenna Regions* to assign layers to antenna regions for identifying the gate and drain regions and to enable antenna calculation. This table is enabled when at least one of the calculate antenna options is active.



4. Use the *Add*, *Edit*, and *Delete* buttons and the G-Spec Builder Form in conjunction with the operators to work with the *Layer Assignment for Antenna Regions* table.

© 2023 Cadence Design Systems, Inc.

5. Select *Use different layer assignments for antenna calculations only* to set layer assignments for antenna extraction different from those set for signal net extraction set on the *Signal* and *Antenna* tabs.



This option activates the *Layer Assignment for Antenna Extraction* table and consequently over-rides any *Layer Assignment for Signal Extraction* option settings.

The columns of this table are described below.

- Specify the *Layer* for which you want to extract antenna data.
- Enter a *Geometry Specification* for each layer through which you want to extract antenna data.

6. Use the *Add*, *Edit*, and *Delete* buttons and the G-Spec Builder Form in conjunction with the operators to work with the *Layer Assignment for Antenna Extraction* table.
7. Select *Signal nets to be excluded from antenna calculation* to specify the nets that should not be considered for antenna extraction and therefore, lets you avoid antenna calculations for the specified pins.

Related Topics

Running Step Extract Form

Calculation of ANTENNAMAXCUTCAR

Antenna Calculation by Abstract Generator

FinFET Support in Abstract Generator

FDSOI Support in Abstract Generator

© 2023 Cadence Design Systems, Inc.

FinFET Support in Abstract Generator

At advanced nodes, in addition to the default antenna data calculation behavior, Abstract Generator allows you to calculate antenna gate area for FinFET devices when the gate area is to be derived from number of fins. A user-defined SKILL hook function `getEffectiveAntennaGateArea` is used for calculating the gate area of the FinFET devices. This SKILL hook function should be defined before running the Extract step.

Syntax:

```
getEffectiveAntennaGateArea (binName gateBBox @rest args)
```

Here, `binName` refers to the bin being used and `gateBBox` is the effective gate area for the specified gate geometry. `@rest args` can include `dbCellViewId`, which is the current database cell view ID in the read mode.

Example:

```
procedure(getEffectiveAntennaGateArea(binName bbox @rest cv)
prog((xa xb ya yb effectiveArea)
      xa = caar(bbox)
      xb = caadr(bbox)
      ya = cadar(bbox)
      yb= cadadr(bbox)

      gateArea = abs((xb-xa)*(yb-ya))
      effectiveArea = gateArea /3
      return(effectiveArea)
);prog
);procedure
```

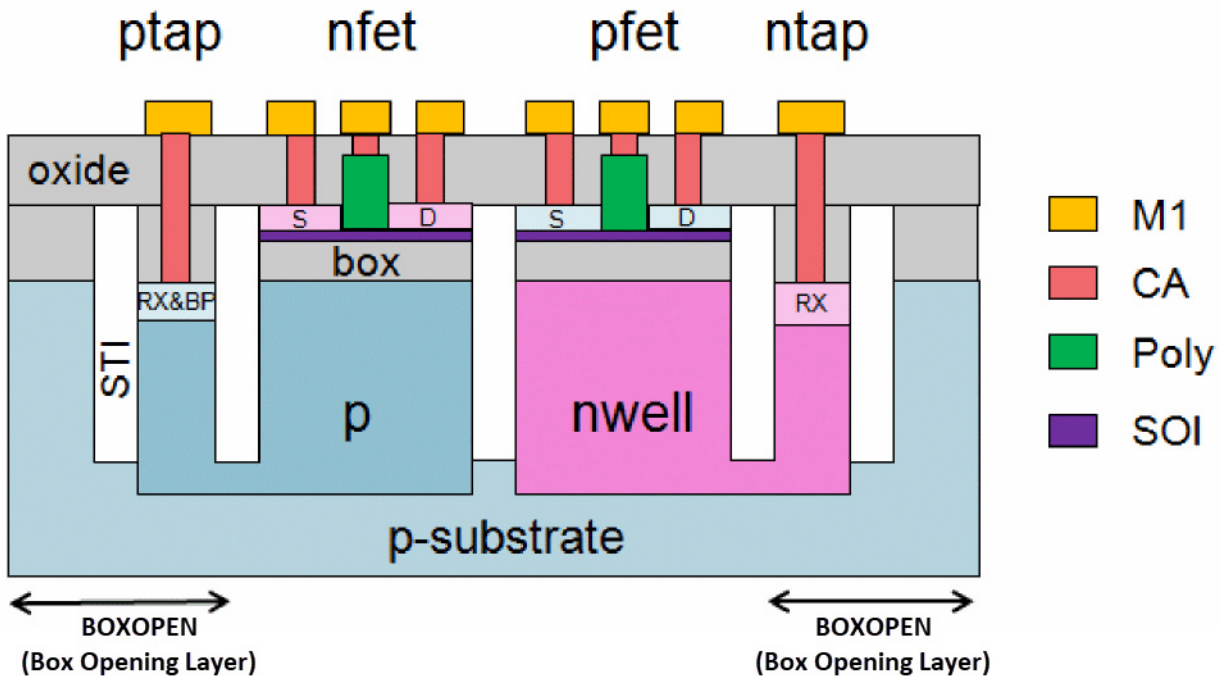
Related Topics

Processing Antenna Information in Standalone Abstract Generator

© 2023 Cadence Design Systems, Inc.

FDSOI Support in Abstract Generator

A Fully Depleted Silicon-On-Insulator (FDSOI) connectivity stack is formed when a thin, fully depleted Silicon channel (`SOI`) and the diffusion (`RX`) are insulated from the substrate and wells by a buried oxide (`box`).



Abstract Generator recognizes the FDSOI process nodes, and for such designs, it includes the FDSOI pseudo-gate area as part of the antenna data calculation.

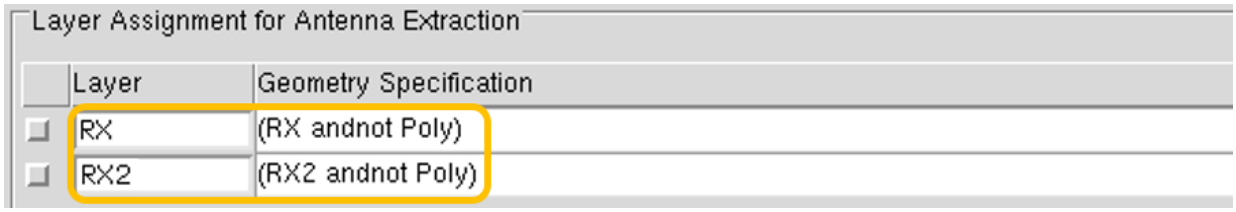
Identification of FDSOI Process Nodes

Abstract Generator checks whether a design is a valid FDSOI process node by evaluating the following:

- The status of the `ExtractAntennaIsSOIProcess` option is set to `TRUE`. The default value is `FALSE`.

© 2023 Cadence Design Systems, Inc.

- The *Geometry Specification* values in the *Layer Assignment for Antenna Extraction* table are valid.



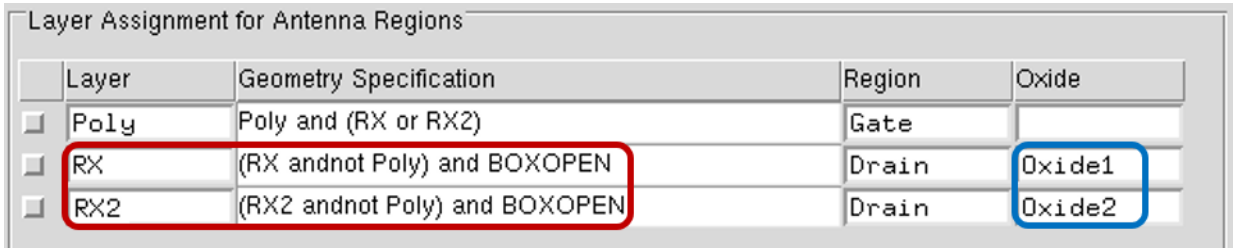
Alternatively, you can use `ExtractAntennaLayers` as shown in the following example.

The geometrical specifications for diffusion layers are set to conducting materials.

```
ExtractAntennaLayers = "(RX ((RX andnot Poly))) (RX2 ((RX2 andnot Poly)))"
```

If the geometrical specification is evaluated as empty, the design is an FDSOI node. If the geometrical specification is evaluated as non-empty, the design is a bulk node.

- The *Geometry Specification* values in the *Layer Assignment for Antenna Regions* table are valid.



Alternatively, you can use `ExtractAntennaDrain` as shown in the following example.

`ExtractAntennaDrain` specifies the geometrical specification for each diffusion layer over bulk.

```
ExtractAntennaDrain = "(RX ((RX andnot Poly) and BOXOPEN)) (RX2((RX2 andnot
Poly) and BOXOPEN))"
ExtractAntennaOxide = "(RX Oxide 1) (RX2 Oxide 2)"
```

The drain geometrical specification identifies the bulk diffusion nodes by inclusion of the box opening layer.

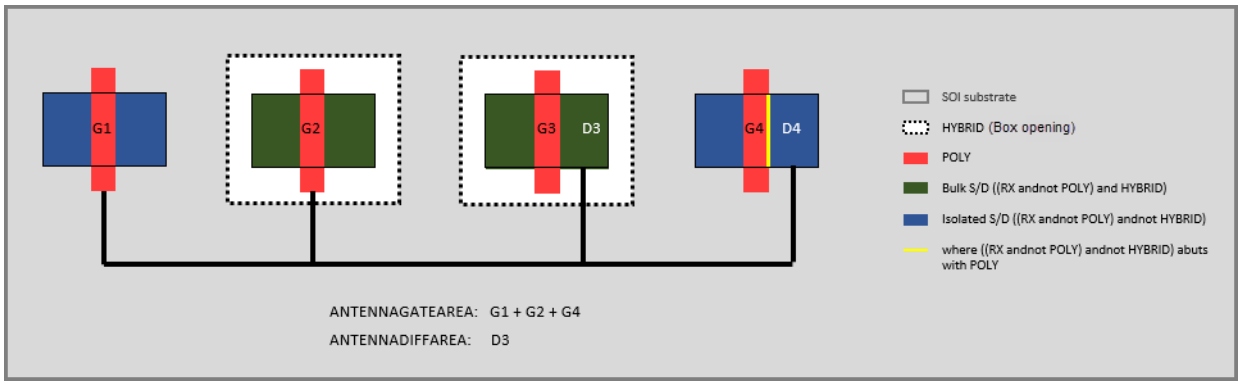
Note: Ensure to specify the correct box opening layer as a part of the `ExtractAntennaDrain` geometrical specifications.

© 2023 Cadence Design Systems, Inc.

FDSOI Pseudo-gate Area Calculation

Pseudo-gates are gate areas characterized by the connectivity of associated adjacent FDSOI diffusion areas with regular unprotected gate areas. If the `ExtractAntennaIsSOIProcess` option is set to `TRUE`, the contribution of an FDSOI pseudo-gate area is included while determining the `ANTENNAGATEAREA`.

The following figure shows the inclusion of the pseudo-gate area during the antenna data calculation.

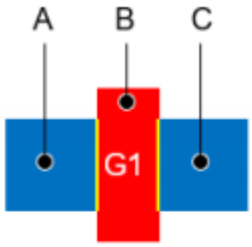
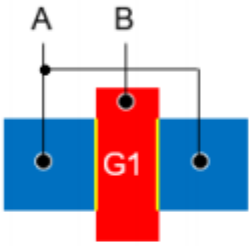
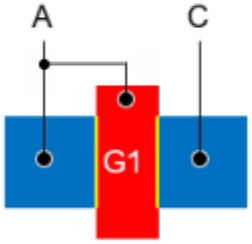
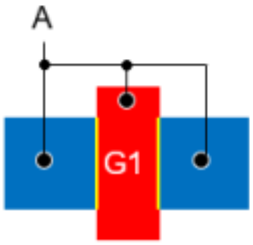


The FDSOI pseudo-gate area contributes to `ANTENNAGATEAREA` and not `ANTENNADIFFAREA`. This is because while transferring charge from diffusion to the adjacent gate, the presence of a buried oxide (insulator) prevents the charge from escaping into the bulk. Abstract Generator recognizes the presence of single and multiple adjacent pseudo-gates and calculates `ANTENNAGATEAREA` accordingly.

During FDSOI pseudo-gate area calculation, the gate area is considered only once if the gate is shared by the two nodes from the same net. The gate area is set to zero if the gate and drain are shorted.

© 2023 Cadence Design Systems, Inc.

Consider nodes A, B, and C. The following table shows how pseudo-gate area is calculated depending on the connectivity of adjacent diffusions to the gate.

Connectivity	node	GATEAREA	Comment
	A B C	G1 G1 G1	pseudo gate regular gate pseudo gate
	A B	G1 G1	pseudo gate to be counted only once ! regular gate
	A C	0 G1	zero gate area since poly/diff "capacitor" is shorted pseudo gate
	A	0	zero gate area since poly/diff "capacitor" is shorted

Technology File Updates

To enable Abstract Generator to calculate antenna data, you need to update the `antennaModels` section of the `foundry` constraint group. You can specify the thickness value in the `techLayerProperties` subsection of the `layerDefinitions` section if antenna side area calculation is to be performed.

© 2023 Cadence Design Systems, Inc.

Abstract Generator supports the calculation of process antenna information for LEF versions 5.4 and above. It cannot compute process antenna information for LEF version 5.3.

Related Topics

FDSOI Connectivity Stack

Requirements for Antenna Calculation

Antenna Model Constraints

© 2023 Cadence Design Systems, Inc.

Antenna Calculation by Abstract Generator

Abstract Generator computes the antenna data for a given layer. The antenna parameters for some of these layers are defined in the LEF technology file. Abstract Generator calculates the following (for metal, cut, and poly layers):

- Antenna gate/diffusion areas
- Antenna metal area/metal side area

Antenna data calculation is complete only after the Abstract step is run, which is the next step in flow after the Extract step. Consequently, the Export LEF Form always takes antenna data from the abstract view rather than from the extracted view.

The computation of antenna data happens in two steps:

1. The main antenna calculation is performed at the end of the Extract step, where Abstract Generator computes antenna data (assuming that all of the extracted geometry is turned into pins in the abstract).
2. During the Abstract step (at the *Adjust* stage), some of the extracted geometry is not turned into pins (for example, if the *Create boundary pins* options for signal and power nets are active), and the process antenna data is removed from layers lower than the lowest pin layer.

For example, assume that you have an extracted net that has *M3* metal, down to *M2* metal, down to *M1* metal, which then connects to a polysilicon gate. This means that there are three computations of `ANTENNAGATEAREA` for *M1*, *M2*, and *M3* (same values) and a further three for `ANTENNAPARTIALMETALAREA` (different values). If, during the *Adjust* stage, *M3* and *M2* become pin shapes but *M1* becomes an obstruction, the computed `ANTENNAMETALAREA` is removed along with the `ANTENNAGATEAREA` for *M1*.

- When a local interconnect layer connects to a polysilicon gate, the local interconnect layer is considered for calculating the `ANTENNAPARTIALMETALAREA` and `ANTENNAMAXAREACAR`. The poly layer is considered for calculating `ANTENNAPARTIALMETALAREA` and `ANTENNAMAXAREACAR`.
- Abstract Generator generates antenna numbers for any terminal direction type of a pin, including `input`, `output`, `inout`, `tristate`, `unused`, `switch`, and `jumper`.

Irrespective of whether the *Extract signal nets* option on the *Signal* tab in the *Extract* step is turned on or off, Abstract Generator generates antenna numbers for a terminal considering all the shapes of that terminal as pin shapes. Also, all the paths from top metal to the gate or drain are considered in calculating the antenna numbers, irrespective of whether the paths contain pin shapes or not.

© 2023 Cadence Design Systems, Inc.

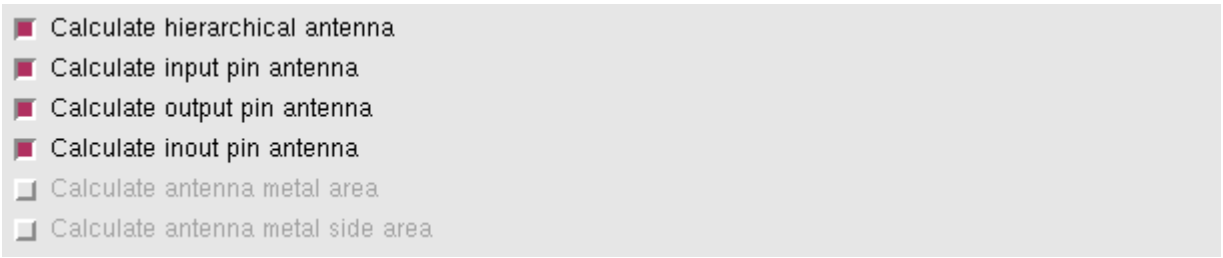
The following options on the *Antenna* tab are used for calculating process antenna data:

- Calculation of ANTENNAMAXCUTCAR:This feature is available through a SKILL function
- *Calculate hierarchical antenna*
- *Calculate input pin antenna*
- *Calculate output pin antenna*
- *Calculate inout pin antenna*
- *Calculate antenna metal area*
- *Calculate antenna metal side area*

OpenAccess supports antenna ratios based on only either side area or non-side area specification in the technology file and not both of them simultaneously. If both side and non-side values are specified in the technology file, the side value overrides the non-side value either when the technology file is updated or when LEF IN is used.

If both *Calculate antenna metal area* and *Calculate antenna metal side area* options are selected, only one of the two antenna metal areas is calculated at any time based on the side and non-side information specified in the technology file. The *Calculate antenna metal area* option becomes active when the non-side area value is specified in the `antennaModels` section of the technology file and the *Calculate antenna metal side area* option remains disabled (grayed out). Similarly, the *Calculate antenna metal side area* option becomes active when the side area value is specified in the `antennaModels` section and the *Calculate antenna metal area* option remains disabled.

As shown in the figure below, because the `antennaModels` section in the technology file contains the non-side area specification, the *Calculate antenna metal area* option is active and can be selected and the *Calculate antenna metal side area* option is disabled.



Related Topics

Running Step Extract Form

© 2023 Cadence Design Systems, Inc.

Processing Antenna Information in Standalone Abstract Generator

Calculation of ANTENNAMAXCUTCAR

LEF/DEF 5.8 Language Reference

© 2023 Cadence Design Systems, Inc.

Calculation of ANTENNAMAXCUTCAR

If you select the *Calculate antenna metal area* option, Abstract Generator also calculates ANTENNAMAXCUTCAR. This value is calculated for the cut layer just above the lowest metal layer on which the pin is present.

Calculation

```
Cut_(i,i+1) CAR = Cut_(i,i+1) PAR + max{ Cut_(i-1,i) CAR,
(Cut_(i,i+1) MAX_CAR(H) - Cut_(i+1) PAR(H)) }
```

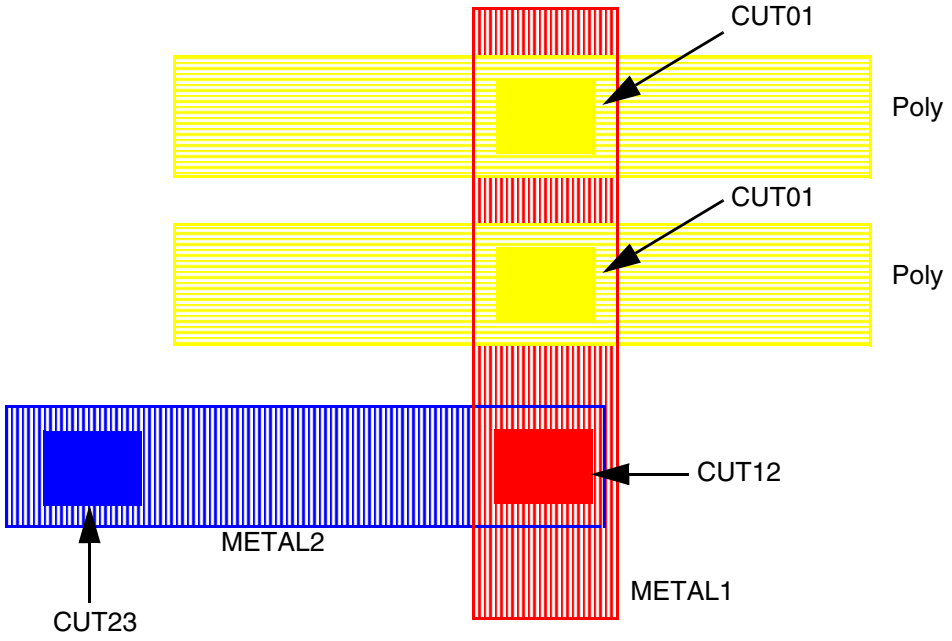
where,

`Cut_(i,i+1)`	Cut layer above the fabricated metal layer
`Cut_(i,i+1) PAR`	Partial cut area divided by gate area under the `i`'th metal layer
`Cut_(i-1,i) CAR`	Cumulative area ratio calculated for the cut layer below
`Cut_(i,i+1) MAX_CAR(H)`	`pcsAntennaMaxCutCAR` value from instance terminal
`Cut_(i,i+1) PAR(H)`	`pcsAntennaPartialCutArea` or `pcsAntennaPartialGateArea` value from instance terminal

By default, the calculation of ANTENNAMAXCUTCAR does not take the CUT01 layer into account. To enable Abstract Generator to consider CUT01 layer while calculating ANTENNAMAXCUTCAR, use the SKILL option ExtractAntennaMaxCutCARLowestCut.

© 2023 Cadence Design Systems, Inc.

Example



Consider `PINA` is on `METAL2` layer. In this case, Abstract Generator will calculate `ANTENNAMAXCUTCAR` for `CUT23` layer as this cut layer is above the lowest metal layer containing the pin. By default, the partial cut areas of `CUT23` and `CUT12` will be considered for `ANTENNAMAXCUTCAR` calculation for `CUT23`. If you set the `ExtractAntennaMaxCutCARLowestCut` option to `true`, `ANTENNAMAXCUTCAR` for `CUT23` calculation will also include the partial cut area of `CUT01`.

```
ANTENNAMAXCUTCAR value CUT23
```

Similarly, if `PINA` is on `METAL1` layer and `ExtractAntennaMaxCutCARLowestCut` is `true`, then `ANTENNAMAXCUTCAR` calculation will be considered for `CUT12` as well as `CUT01` layers.

```
ANTENNAMAXCUTCAR value CUT01
ANTENNAMAXCUTCAR value CUT12
```

If the `ExtractAntennaMaxCutCARLowestCut` option is `true`, `ANTENNAMAXCUTCAR` value is displayed for `CUT01` only if the pin is on `METAL1`.

Related Topics

Running Step Extract Form

© 2023 Cadence Design Systems, Inc.

Processing Antenna Information in Standalone Abstract Generator

Antenna Calculation by Abstract Generator

© 2023 Cadence Design Systems, Inc.

Keywords Used in Antenna Calculation

The following table describes antenna keywords related to per layer library data.

Keyword	Description
`Area`	The area of metal wires connected to the polysilicon gate: width(W) x length(L).
`SideArea gate`	The side wall area of metal wires connected to the polysilicon gate: 2(W+L) x thickness.
`Diff gate`	Top mask layer or any metal layer connecting between polysilicon and output diode.
`Cum polysilicon`	Any lower metal layer that connects directly to the gate(s) is also cumulated to the antenna calculation on the metal layer currently being checked.
`PWL variable`	Piecewise linear format. This is used to define the diffusion antenna ratio.
`PAR`	Partial antenna ratio.
`CAR`	Cumulated antenna ratio.
`PAE`	Process antenna effect.
`DIFFUSEONLY`	Keyword to specify the current antenna factor to only be used when the corresponding layer is connected to diffusion.

Related Topics

Running Step Extract Form

Processing Antenna Information in Standalone Abstract Generator

Antenna Calculation by Abstract Generator

© 2023 Cadence Design Systems, Inc.

Checks Performed During Antenna Calculation

If you activate antenna extraction by selecting any *Calculate* option, Abstract Generator checks for the following:

- Extraction of poly silicon layer
- Extraction of diffusion layer
- Specification of diffusion layer in terms of

  `(Diff (Diff andnot Poly))`

If any of the above checks fail, the following corresponding error messages will be displayed:

- Failure of extraction of polysilicon layer generates the following error message:

  ```
  ABS-606: Cell name: layer POLY is not listed in any of the layer
  assignment for extraction tables. This is required for antenna
  extraction.
  ```

- Failure of extraction of diffusion layer generates the following error message:

  ```
  ABS-607: Cell name: layer DIFF is not listed in any of the layer
  assignment for extraction tables. This is required for antenna
  extraction.
  ```

- Missing specification of diffusion layer in terms of `(Diff (Diff andnot Poly))`

  ```
  ABS-608: Cell name: layer DIFF must be specified in one of the
  layer assignment for extraction tables as type (diff diff andnot
  poly) for full antenna extraction to take place.
  ```

- Failed to extract because the number of shapes that were extracted for net name exceeds the current extraction limit of value.

  (ABS-13009) Use the `ExtractShapeLimit` option to increase the extraction limit and rerun the command.

  This situation might result because of one of the following reasons:

  - The number of shapes being extracted from the current design exceeds the maximum permitted value. The default value is 10000.
  - The net has the wrong terminal type set.
  - The design contains a short between nets.

  You can take one of the following actions to overcome this error:

© 2023 Cadence Design Systems, Inc.

- Increase the extract shape limit by using the following setting either in the GUI or in the .abstract.options file:

```
absSetOption( "ExtractShapeLimit" "<any positive number>" )
```

For example,

```
absSetOption( "ExtractShapeLimit" "400500" )
```

- Additionally, you can set the stack size for the abstract process to unlimited by typing the following in the terminal window before invoking Abstract Generator:

```
limit stacksize unlimited
```

- Select *Cells – Terminal Properties* to open the Terminal Properties form and check that the terminal type is set correctly for the net in question.

■ (ABS-13010) Cell "%s": Extraction has been aborted because a short has been detected between the nets *name* and *name*.

Check the following:

- If you are performing antenna extraction, ensure that the *Layer Assignment for Antenna Extraction* table on the *Antenna* tab has been populated correctly. The geometry specification for the diffusion layer in the table should read 'diff diff and not poly'.
- Ensure that the *Layer connectivity* on the *General* tab is set up correctly for all layers, especially for polysilicon and metal1 and diffusion and metal1, such as '(poly metal1 contact) (diff metal1 contact)'.

■ (ABS-13020) Cell *name*: matching of the gate region with correct antenna oxide type has failed. This type of failure occurs when the abstract generator is unable to identify the diffusion layer for gate calculation. Check the settings in the *Layer Assignment for Antenna Regions* table on the *Antenna* tab.

Abstract Generator is unable to identify which diffusion type a gate is over. This can occur if you specify the antenna regions like the one below.

Layer	Geometry Specification	Region	Oxide
poly	*poly* and (*diff* or *diff2*)	Gate	
diff	*diff* andnot *poly*	Drain	*Oxide1*

Where *poly*, *diff*, *diff2* and *Oxide1* are replaced by the real names of the layers in question.

Any *Oxide* option can be set here (or even none), but the antenna style must be LEF 5.5.

© 2023 Cadence Design Systems, Inc.

In the above example *diff2* has not been specified as a drain region but has been specified in the *Geometry Specification*. This means that any gates over *diff2* will not be identified with a particular oxide.

The fix for this example is to add the following extra line:

```
Layer      Geometry Specification      Region      Oxide
-----      ----------------------      ------      -----
poly       poly and (diff or diff2)    Gate
diff       diff andnot poly            Drain       Oxide1
diff2      diff2 andnot poly1          Drain       Oxide2
```

Another scenario that can cause this message to appear, is where there is a complicated size operator specified in the *Geometry Specification* for the polysilicon layer. If you have then sized up the diffusion layer in the *Gate* specification, it is possible that this could create a gate area which would not overlap the unsized diffusion layer.

To avoid receiving the above extraction-related error messages, ensure that you have entered a valid expression in the *Layer Assignment for Signal Extraction* table on the *Signal* tab and *Layer Assignment for Antenna Extraction* table on the *Antenna* tab. For example, you can specify the following in these tables:

Layer	Geometry Specification
Poly1	Poly1
CONT	CONT
NDIFF	NDIFF andnot Poly1

Related Topics

Running Step Extract Form

Processing Antenna Information in Standalone Abstract Generator

Antenna Calculation by Abstract Generator

© 2023 Cadence Design Systems, Inc.

Comparison of Geometry Specifications for Antenna Regions and Antenna Extraction

When defining the geometry specification for the polysilicon layer in the *Layer Assignment for Antenna Extraction* table, do not use the same specification as the one you used in the *Layer Assignment for Antenna Regions* table. If you do, Abstract Generator cannot extract the full polysilicon shape and cannot output antenna values for the input gate.

In the following example, notice the different geometry specifications in the two tables.

Layer Assignment Table	Geometry Specification
Antenna Regions	`(polysilicon and (pdiff or ndiff)`
Antenna Extraction	`poly`

Excluding Signal Nets from Antenna Calculation

The *Signal nets to be excluded from antenna calculation* field allows you to specify the nets that should not be considered for antenna extraction and therefore, lets you avoid antenna calculations for the specified pins. For example, by specifying the expression as `in*` for the signal nets to be excluded, you can exclude all nets with names beginning with `in` from antenna calculation.

This field is enabled only if one of the calculate antenna options is selected.

Related Topics

Running Step Extract Form

Processing Antenna Information in Standalone Abstract Generator

FinFET Support in Abstract Generator

FDSOI Support in Abstract Generator

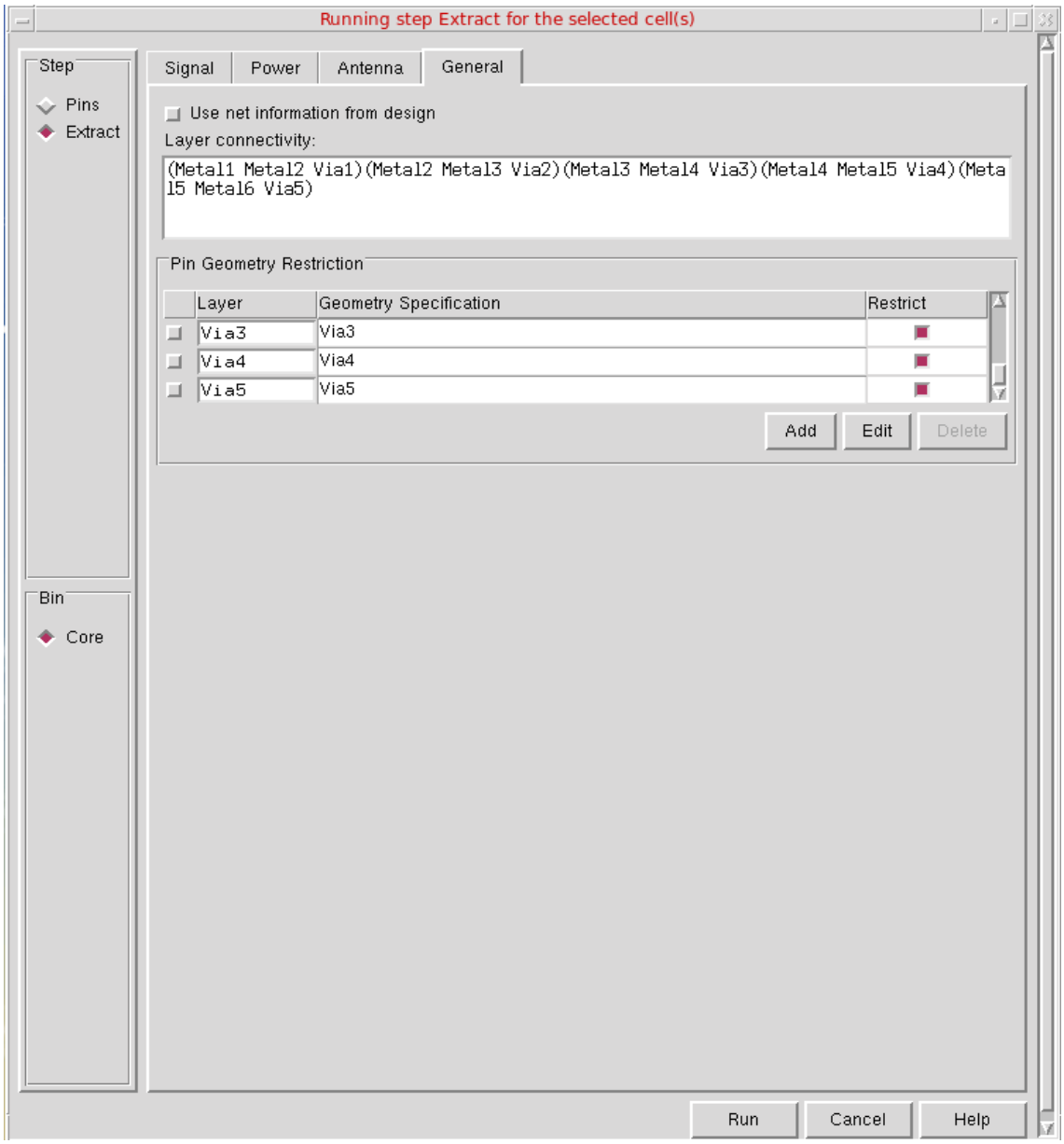
© 2023 Cadence Design Systems, Inc.

Specifying Connectivity Information in Designs

You can use the Running step Extract form to extract geometries on signal or power nets or to calculate process antenna information. You can specify connectivity information in a design using the *General* tab in the Running step Extract form.

To specify options in the *General* tab in the Running step Extract form:

1. Choose *Flow – Extract* and click the *General* tab.

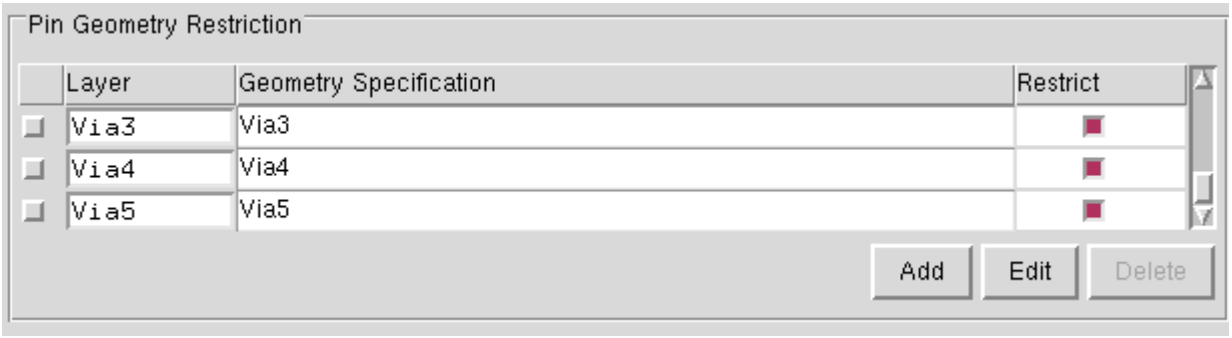


© 2023 Cadence Design Systems, Inc.

2. Select *Use net information from design* if your design already contains connectivity information. If you select this option, Abstract Generator skips the extraction based on overlapping shapes and use the existing connectivity information in your design to create nets.

3. Use the *Layer connectivity* field to specify the connectivity between layers. This allows you to specify the layer connectivity without having to add via definitions in the technology file.

   The *Pin Geometry Restriction* table provides you with finer control over final pin geometry.



4. From a list of all the layers specified in the technology file, select those that you want to assign for pin geometry.

5. Enter the *Geometry Specification* for the regions where pins are either to be restricted to or avoided.

6. Choose whether pins are to be allowed or avoided (*Allow* or *Avoid*) in the geometry specified in the *Geometry Specification* field. The specified geometry specification (G-spec) describes the region where pins are to be allowed or avoided.

7. In the *Restrict* column, select or deselect the button for adjusting pins and not for deleting the existing pins, which can slow down the performance of abstraction process. The red button indicates that the option is selected and the white button indicates that the option is not selected. To create the required pins, specify the geometry specification for the pin layers during the Extract step.

   By default, the *Restrict* column is used for both signal and power pins. You can choose whether only signal or power pins are to be restricted to the specified region or avoided. To do this:

   a. Set the following Shell environment variable before starting the Virtuoso session:

      ```
      setenv AllowAvoidSignalAndPowerPinsSeparately
      ```


© 2023 Cadence Design Systems, Inc.

b. Set the following options of the `absAbstract` SKILL function to specify only power pins.

```
absSetBinOption("Core" "AbstractAdjustAllowPowerPin" "(Metal1 (Metal1
andnot prBoundary boundary size -1)) ")
absSetBinOption("Core" "AbstractAdjustAvoidPowerPin" "(Metal2 (Metal2
andnot prBoundary boundary size -1)) ")
```

c. Set the following options of the `absAbstract` SKILL function to specify only signal pins.

```
absSetBinOption("Block" "AbstractAdjustAllowPin" "(Metal1 (Metal1 andnot
prBoundary boundary size -0.05)) ")
absSetBinOption("Block" "AbstractAdjustAvoidPin" "(Metal1 (Metal1 andnot
prBoundary boundary size -0.05)) ")
```

If the `AllowAvoidSignalAndPowerPinsSeparately` environment variable is not set, the `AbstractAdjustAllowPin` and `AbstractAdjustAvoidPin` options are used to specify both signal and power pins.

8. Select the *Add*, *Edit*, and *Delete* buttons and the G-Spec Builder Form in conjunction with the operators to work with the *Layer Assignment for Pin Geometry* table.

Related Topics

Running Step Extract Form

© 2023 Cadence Design Systems, Inc.

Restrictions for Signal and Power Pin Geometry

The following restrictions apply to setting signal and power pin geometry:

- If there is a `metal0` local interconnect layer, it would have to be entered manually.
- If two layers connect by overlap, specify a layer pair `(M0 M1)`.
- It is not permitted to specify `(layer1 layer2 via12)` and `(layer1 layer2)` and so on at the same time.

For example,

```
(metal1 metal2 via12) (metal2 metal3 via23)
(poly metal1 cont)
(pdiff metal1 cont)
(ndiff metal1 cont)
(poly li)
(ndiff li)
(pdiff li)
```

This means,

- `metal1` connects to `metal2` through `via12` layer
- `metal2` connects to `metal3` through `via23` layer
- `poly` connects to `metal1` through `cont` layer
- `pdiff` connects to `metal1` through `cont` layer
- `ndiff` connects to `metal1` through `cont` layer
- `poly` connects to local interconnect by overlap
- `ndiff` connects to local interconnect by overlap
- `pdiff` connects to local interconnect by overlap

Related Topics

Running Step Extract Form

Specifying Connectivity Information in Designs

© 2023 Cadence Design Systems, Inc.

Troubleshoot Extract Issues

You can face the following issues while using the Extract step in the abstract generation process.

ABS-13006

Message: Cell *name*: layer 'POLY' is not listed in the *Layer Assignment for name Extraction* table. This layer is required for antenna extraction. Select 'POLY' in the *Layer* column and specify other information for this layer to be extracted. See Extracting Power Nets in Standalone Abstract Generator and Specifying Layers and Geometries for Extraction.

ABS-13007

Message: Cell *name*: layer 'DIFF', which is required for antenna extraction, is not listed in the *Layer Assignment for name Extraction* table. For antenna extraction, select 'DIFF' in the *Layer* column and specify the layer 'DIFF' as type '(DIFF andnot POLY)', where DIFF and POLY are the diffusion and polysilicon layers in question. See Specifying Layers and Geometries for Extraction and Specifying Layers and Geometries for Extraction.

ABS-13008

Message: Cell *name*: either the layer 'DIFF' is not specified as type '(diff diff andnot poly)' in the *Layer Assignment for name Extraction* table or the geometry specification for a layer contains two or more layers from the *Layer* column. Select 'DIFF' in the *Layer* column and specify it as type '(diff diff andnot poly)', where diff and poly are to be replaced with the real names of the diffusion and polysilicon layers in question. If a geometry specification uses a temporary layer to limit extraction and the temporary layer is a combination of two or more layers from the *Layer* column, do not select the individual layers in the *Layer* column.

© 2023 Cadence Design Systems, Inc.

Description: This message appears when a diffusion layer is not specified in one of the layer assignment for extraction tables. This message might also appear when a geometry specification contains two or more layers from the *Layer* column. See Extracting Power Nets in Standalone Abstract Generator and Layer Assignment Tables.

ABS-13012

Message: Cell `name`: a possible must join terminal *name* has been ignored because the terminal was not listed in the *Create Must Connect Pins* option on the *name* tab. Ensure that all must connects are joined for a specific terminal, either by activating the *Always* option on the *Signal* tab or listing the terminals in the *Only on terminals named* field.

ABS-13014

Message: Cell `name`: there is no pin geometry on the net `name`. To create pins on a specific layer, select the *Create Pins* check box corresponding to the layer in the *Layer Assignment for* `name` *Extraction* table.

Description: It is usually unnecessary to create pins on polysilicon, contact, or diffusion layers. See Specifying Layers and Geometries for Extraction.

ABS-13018

Message: Cell `name`: net `name` process antenna calculations for terminal `name` on instance `name` have not been used.

Description: Abstract Generator has failed to identify any extracted nodes as representing instance terminal pin shapes. Because the nodes have not been matched to an instance pin, the associated antenna calculations have not been taken into account. This problem could be a result of the net not being fully routed in DEF.

ABS-13019

© 2023 Cadence Design Systems, Inc.

Message: Cell *name*: net *name* process antenna *type* value has been ignored because it is different from other *layerName* values for terminal *name* on instance *name*. This might indicate that the antenna information stored on a terminal is inconsistent with itself. This could occur if there is an error in the antenna calculation or the property values have changed in the abstract. The inconsistency might also indicate that the pin area for the instance terminal has changed since it was first subtracted in the Adjust step.

ABS-13021

Message: Cell *name*: a short has been detected between the nets *netName* and *netName* during antenna calculation. This indicates a problem in the design unless the nets are connected by a polysilicon resistor. Though the short, if it exists between the nets, is ignored for antenna calculation but is considered for the signal net extraction because either the *Use different layer assignments for antenna calculations only* option is selected or the extraction depth has been limited.

Troubleshooting: When you are performing antenna extraction in analog blocks, specify `polysilicon andnot resistor` in the *Layer Assignment for Signal Extraction* and *Layer Assignment for Power Extraction* tables. Remove the resistor part in the *Layer Assignment for Antenna Extraction* table.

ABS-13022

Message: Cell *name*: no pin shape will be created on the layer *name* because the layer does not have net purpose defined in the technology data. Ensure that the technology file is correct. Otherwise, check the design.

Troubleshooting: Use the Technology File Manager to define the missing LPPs in the `techLayerPurposePriorities` section of the technology file.

Related Topics

Specifying Layers and Geometries for Extraction

© 2023 Cadence Design Systems, Inc.

Extracting Power Nets in Standalone Abstract Generator

Processing Antenna Information in Standalone Abstract Generator

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

5

Abstract Step in Standalone Abstract Generation

The Abstract step describes the third stage of the abstract generation process. Abstract Generator adjusts pin shapes, performs blockage modeling, creates sites, and calculates overlap layers during the Abstract step.

To run the Abstract step:

1. Select one or more cells in the main window and click the *Abstract* button. Alternatively, select *Flow – Abstract*.

   This opens the Running step Abstract form.

   There are six tabs on the Running step Abstract form- *Adjust*, *Blockage*, *Density*, *Fracture, Site*, and *Overlap*.

2. You can select the options you need on this form and click the *Run* button.

   The *Percent Complete* message box reports the progress of the step.

If there are no valid Pins or Extract views for a cell, Abstract Generator automatically runs the Pins and Extract steps prior to running the Abstract step.

The *QuickAbstract* option enables you to improve the performance of Abstract Generator by 30 percent for small and medium sized designs. If this option is set to true, Abstract Generator does not generate intermediate cellviews namely Pin and Extract cellviews; it generates only Abstract cellviews. By default, the *QuickAbstract* option is set to false.

When the Abstract step is complete, Abstract Generator places a symbol against the cell in the *Cell* pane to indicate the status of the Abstract view that was created. Abstract Generator automatically detects shapes placed in WSP regions and ignores these shapes while creating blockages.

In the Running step Abstract form, use the radio buttons on the left to navigate the options pages for the different steps and bins you are using.

© 2023 Cadence Design Systems, Inc.

Multi-Patterning Technology

To support Multi-Patterning Technology (MPT), the options on the various tabs of the Running step Abstract form now recognize the colors of the original shapes in the layout editor. Therefore, the shapes created in the abstract inherit the colors from the original shapes. For example, two pin shapes created by fracturing a single rectilinear shape will inherit the color of the original rectilinear shape. Abstract Generator respects color shifts and layers that are present on instances or shapes.

Related Topics

Pins Step in Standalone Abstract Generation

Extract Step in Standalone Abstract Generation

Multi-Patterning Technology Support in Abstract Generator

Retain Layout Information in the Abstract View

Creation of Fill Blockages on Specific Layers

Bus Annotation in Abstract Generator

Layout Dual View Support

Support for Copying Mixed Signal Routing Constraints from Layout to Abstract View

Suppress Messages Issued by Abstract Generator

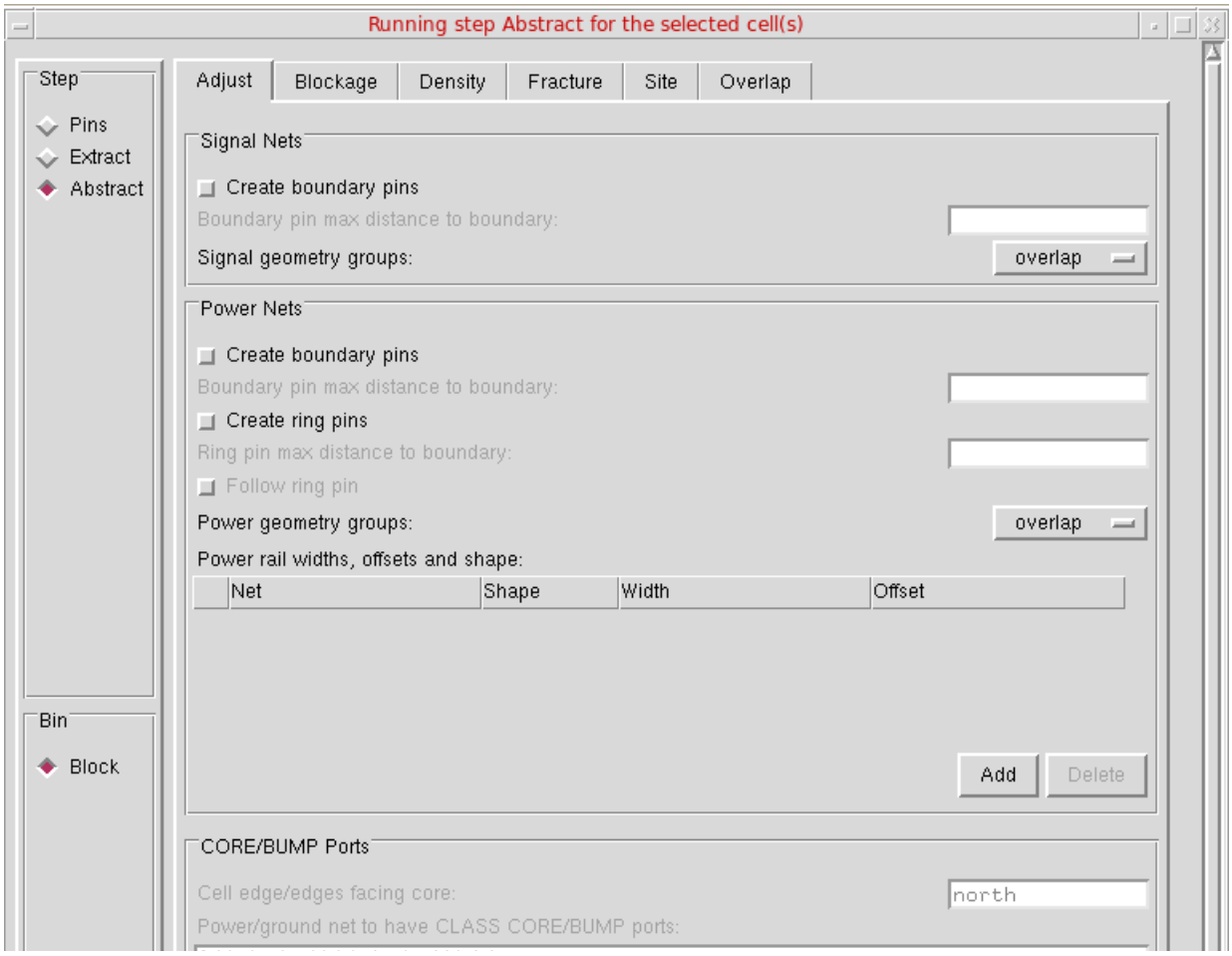
© 2023 Cadence Design Systems, Inc.

Customizing Pin Shapes in Standalone Abstract Generator

The options on the *Adjust* tab of the Running step Abstract for the selected cell(s) form control how pin shapes are adjusted during connectivity extraction for creating the final required pin shapes. These pin shapes are then fractured into rectangles. The options help you ensure that if a pin shape is reduced in any way, the discarded part of the pin is turned into a blockage, and the blockage is spaced sufficiently away from the reduced pin shape.

To customize the pin shapes in Standalone Abstract Generator:

1. Choose *Flow – Abstract*. The *Adjust* tab is the first tab in the Running step Abstract for the selected cell(s) form.

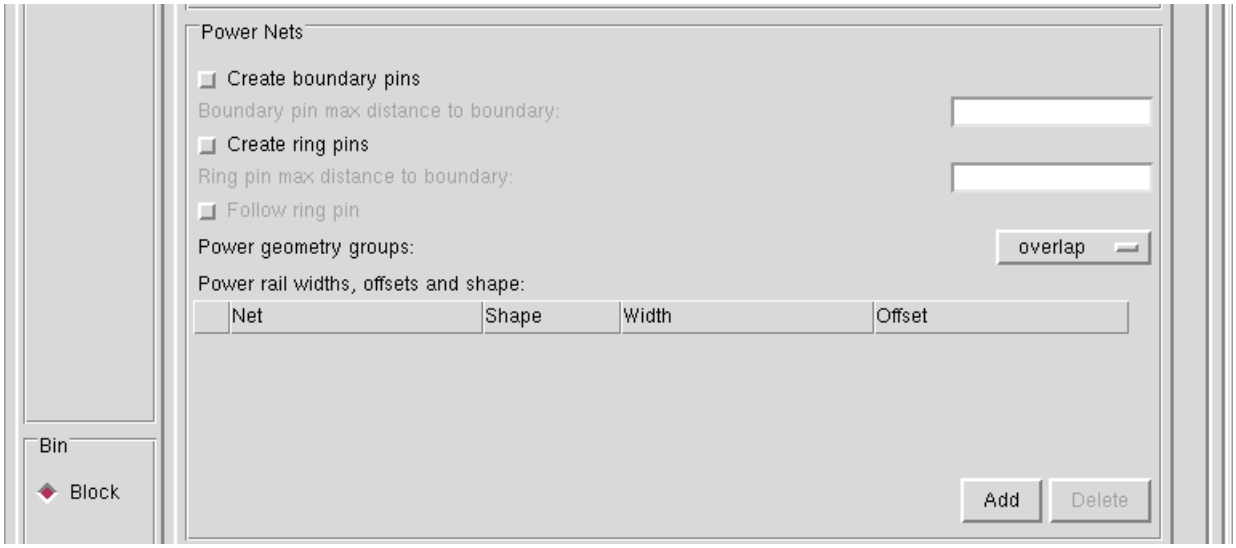


2. In the *Signal Nets* section, you can control how pin shapes are adjusted during connectivity extraction to create the final required pin shapes for signal nets.

© 2023 Cadence Design Systems, Inc.

- Select *Create boundary pins* to adjust the pin shapes created during extraction to be square and on the cell boundary.
- Use *Boundary pin max distance to boundary* to specify the maximum distance (in microns) of the signal net or power net geometries from the boundary so that they can be considered as boundary pins.
- Select *Signal geometry groups* to control how signal net shapes are grouped into LEF PORTS.

3. In the *Power Nets* section, you can control how pin shapes are adjusted during connectivity extraction to create the final required pin shapes for power nets.

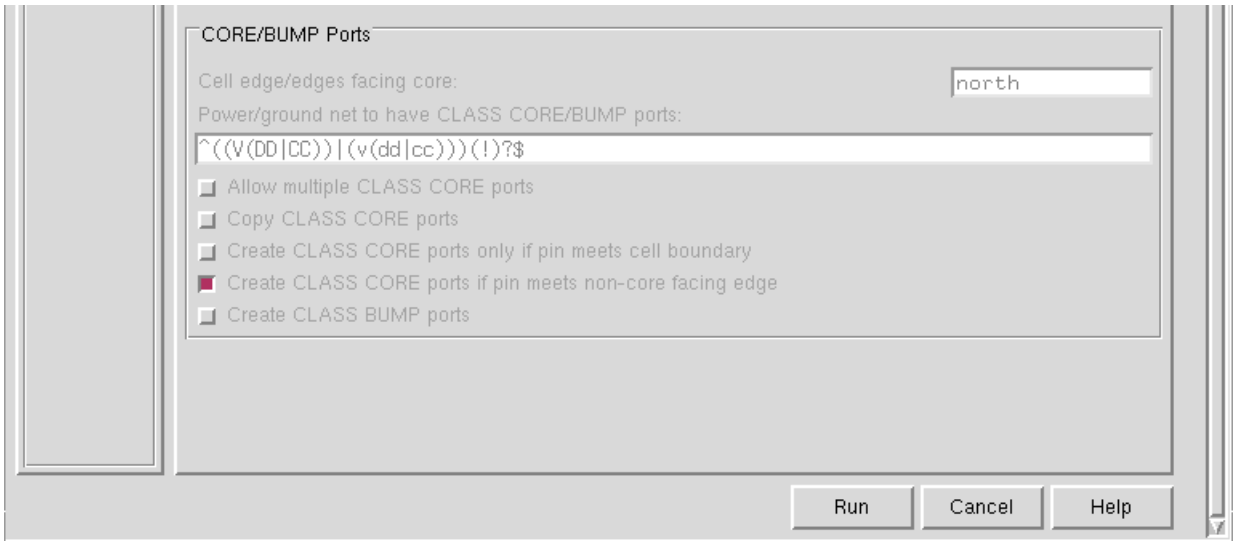


- Select *Create boundary pins* to adjust the pin shapes created during extraction to be square and on the cell boundary.
- Use *Boundary pin max distance to boundary* to specify the maximum distance (in microns) of the signal net or power net geometries from the boundary so that they can be considered as boundary pins.
- Select *Create ring pins* to create ring pins where any extreme shapes, that are close to the cell boundary existing in a power net are found to form a ring.
- Use *Ring pin max distance to boundary* to specify (in microns) the maximum distance within which the extracted power net geometry can be from the boundary in order for it to be considered as part of the ring, and therefore, be included in the abstract as pin geometry.
- Select *Follow ring pin* to consider all pin shapes, including the ones that are not part of the ring, as pins in the final abstract.

© 2023 Cadence Design Systems, Inc.

- Select *Power geometry groups* to control how power shapes are grouped into LEF `PORTS`.
- Use *Power rail widths, offsets and shape* to specify the characteristics of the power and ground rails for the current bin.

4. In the *CORE/BUMP Ports* section, you can specify the following CLASS CORE-specific information to Abstract Generator.



- Use *Cell edge facing core* to specify the direction in which the `CORE`-facing edge is pointing, *north*, *south*, *east*, *west,* or a combination of any of the four core edges.
- Use *Power/ground net to have CLASS CORE ports* to specify the nets that should have `CLASS CORE` ports. This can be specified by entering a regular expression in the field.
- Select *Allow multiple CLASS CORE ports* to specify that two ports equidistant from the core-facing edge of a pad are both assigned the `CLASS CORE` statement in LEF.
- Select *Copy CLASS CORE ports* to duplicate any geometry found to belong to a `CLASS CORE` port on the net (set by using the *Power/ground net to have CLASS CORE ports* option) in a non-`CLASS CORE` port to the exported abstract LEF.
- Select *Create CLASS CORE ports only if pin meets cell boundary* to specify that only the pins that either touch or cross the core-facing edge of the boundary become `CLASS CORE` ports in LEF.

© 2023 Cadence Design Systems, Inc.

- Select *Create CLASS CORE ports if pin meets non-core facing edge* to allow the pins that touch the non-core facing edge to become eligible to be included as CLASS CORE ports.
- Select *Create CLASS BUMP ports* to consider power nets of cell type IO for creating the CLASS BUMP ports.

5. Click *Run*.

Related Topics

Running step Abstract Form

© 2023 Cadence Design Systems, Inc.

Square and Non-Square Boundary Pins

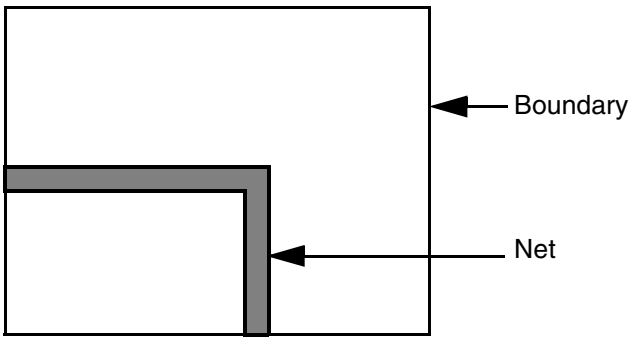
The *Create boundary pins* option exists at two locations on the *Adjust* tab — *Signal Nets* and *Power Nets* groups. By default, this option is off for all bins except the *Block* bin and is disabled and off for *Core* bin.

Square Boundary Pins

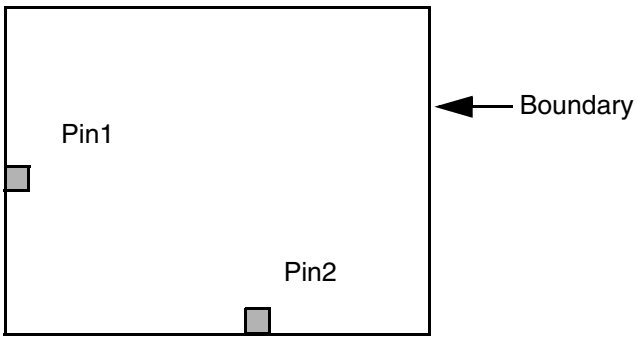
The *Create boundary pins* option lets you adjust the pin shapes created during extraction to be square and on the cell boundary. This can be useful when processing blocks where pin geometry extends deep inside the block after extraction. If the *Create boundary pins* option is turned off, the pin will retain the shape generated by the Extract step, which can still be modified using the *Layer Assignment for Signal Extraction* table options.

The method used by Innovus to check width-dependent spacing rules might lead to false violations, so you should restrict the pin geometry to be a square near the boundary of the block by using the *Create boundary pins* option.

Consider the following net inside a block.



With the *Create boundary pins* option turned on, this becomes:



© 2023 Cadence Design Systems, Inc.

The pins that originally extended inside the block are transformed into squares of the same dimension as the original width of the pin.

If the area of the metal wire from which a boundary pin is cut is less than the minArea constraint defined in the technology file for that layer, a warning message indicating the minimum area rule violation is displayed.

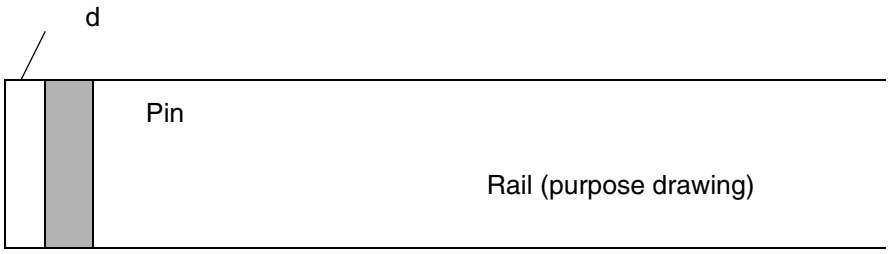
When verifying the boundary pin size, Abstract Generator does not calculate the metal area of the square boundary pin generated in the abstract. Instead, it calculates the area of the metal wire from which the boundary pin is cut. If the area of the metal wire is less than the minArea constraint, a warning message is generated indicating that the minimum area rule is violated.

Non-Square Boundary Pins

Sometimes, the resultant pin shape might not be square even when *Create boundary pins* is turned on. This can happen when you do not run any extraction but instead use preset pins. Abstract Generator might need to stretch the pin because it might not be positioned right at the edge of the underlying geometry. Abstract Generator stretches pins only up to the PR boundary and not beyond it.

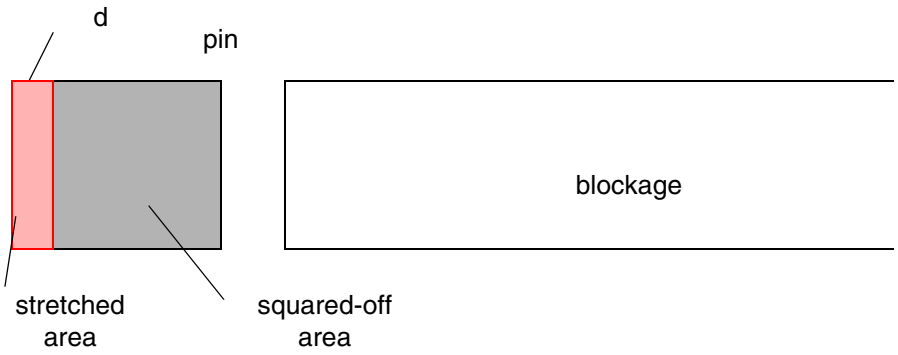
Example

Consider the following figure.



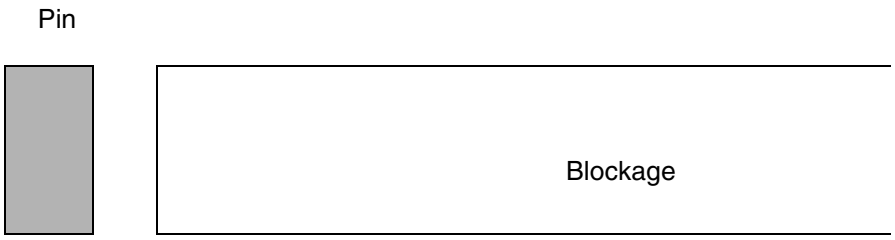
© 2023 Cadence Design Systems, Inc.

The pin does not extend right to the edge of the underlying geometry but the distance ‘d’ is less than the minimum allowed: a separation plus the minimum wire width.



When the *Create boundary pins* option is selected, Abstract Generator first squares off the pin to the right, maintaining the existing left boundary and adjusting the right boundary (the darker hatched area). Then, because distance ‘d’ is too small to accommodate a visible spaced blockage, Abstract Generator stretches the pin back to the boundary (the lighter hatched area).

When the *Create boundary pins* option is not selected, the pin is not squared off first, and the right-most edge is the same as that of the original pin shape. Abstract Generator stretches the pin to the left edge of the underlying geometry as shown in the figure below.



Related Topics

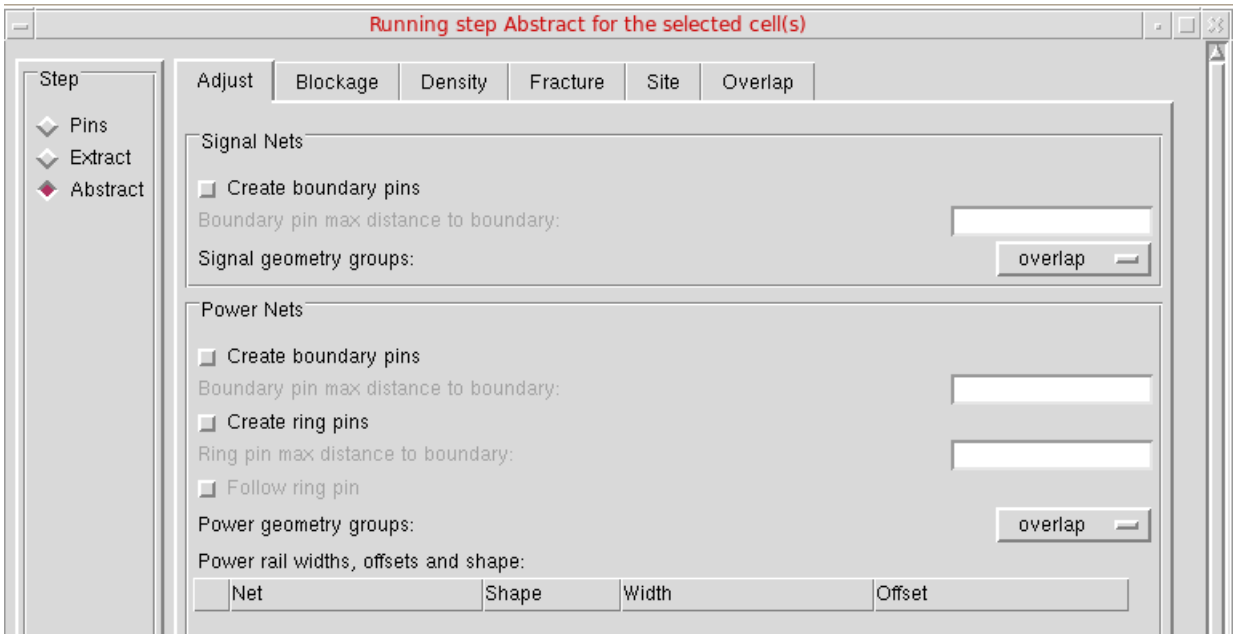
Automatic Pin Stretching

© 2023 Cadence Design Systems, Inc.

Specifying Power and Ground Rail Characteristics

To specify the characteristics of the power and ground rails for the current bin:

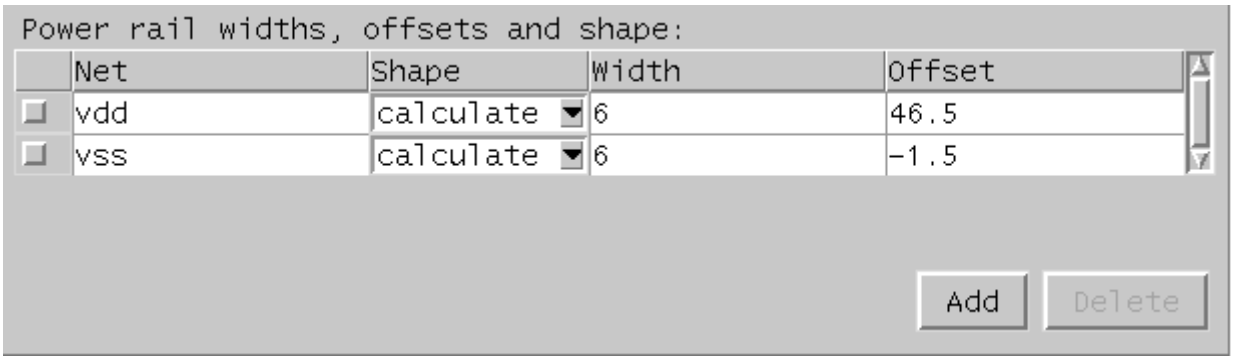
1. Choose *Flow – Abstract*. The *Adjust* tab is the first tab in the Running step Abstract form.



2. In the *Signal Nets* section, you can control how pin shapes are adjusted during connectivity extraction to create the final required pin shapes for signal nets.
3. In the *Power Nets* section, you can control how pin shapes are adjusted during connectivity extraction to create the final required pin shapes for power nets.
4. Use *Power rail widths, offsets and shape* to specify the characteristics of the power and ground rails for the current bin.
5. Open the *Power rail widths, offsets and shape* table.

© 2023 Cadence Design Systems, Inc.

The table contains a separate row for each power rail.



The *Power rail widths, offsets and shape* option will be unavailable if the *Create ring pins* option is selected. The *Create ring pins* option is available only if you are processing a block.

Note: If you try to process a block in the Core bin, the Power rail widths, offsets and shape option will lead to a long run time. To avoid this, process blocks in the Block bin.

6. Use *Net* to specify the net name in this column.
7. Select *Shape* to select from *feedthru*, *abutment*, *calculate* and *none*.
8. Use *Width* to specify the rail width (in μ) in this column.
9. Use *Offset* to specify the rail position (in μ) in this column.
10. You can insert or delete rows from the table by selecting the appropriate button, *Add* or *Delete*.
11. Check the box to the left of a row to mark it for deletion. If the table is left empty, Abstract Generator will try to determine the rail data for the *Core* bin, including calculating the rail and its shape. For *Block* and *IO* bins, no calculations are performed if the table is left empty. This is the default situation.

The table below describes the impact and resultant of selecting one of these values in the *Shape* column.

© 2023 Cadence Design Systems, Inc.

Net *Shape*	Implication	Default Setting
feedthru	The *Width* and *Offset* values do not need to be set because no calculation is performed. The terminal associated with that net will be given this shape property	Available for *IO*, *Corner*, and *Block* bins.
abutment		
calculate	The *Width* and *Offset* values must be entered for ALL nets because all power rail nets must be calculated.	Available only for cells in the *Core* bin.
none	No calculation will be performed.	Available for *IO*, *Corner*, and *Block* bins.

You cannot set the same power rail net more than once with different *Shape* values associated with it.

If the *Power rail widths, offsets and shape* option is set to *feedthru, abutment* or *none,* Abstract Generator will only use the specified values to set terminal properties. That is, if only one power rail net is entered into the table, that will be the only rail used to set terminal properties.

If Abstract Generator establishes that a rail can be represented by either a feedthru or an abutment rail, it will always choose the abutment shape, as shown below.

```
Abstract: 2001/10/31 14:51:52:

Info:     Power Rail Analysis:

================================================================================
Rails found
================================================================================
rail  type    net  layer   width  offset  shape     direction   cells
--------------------------------------------------------------------------------
1     power   vdd  met1    2.5    10.87   abutment  horizontal  all
2     power   vdd  met1    5.645  7.725   feedthru  horizontal  all
3     ground  vss  met1    2.8    1.51    abutment  horizontal  all
4     ground  vss  met1    3.965  1.51    feedthru  horizontal  all

Abstract: 2001/10/31 14:51:52:

Info: Using power rails 1 and 3 to set terminal properties.These settings can be
altered using the Adjust “Power rail widths and offsets”  table.
```


© 2023 Cadence Design Systems, Inc.

Related Topics

Running step Abstract Form

Customizing Pin Shapes in Standalone Abstract Generator

© 2023 Cadence Design Systems, Inc.

Power Rail Scenarios

To establish a better understanding of how this functionality operates, consider the following scenarios:

- **Rail data is not specified**

  Example 1: In this scenario, no rail data has been specified and the power rail analyzer has attempted to find rails in each of the cells.

  At the end of the analysis phase, a table is produced listing the rails found and the cells that contain them. Also listed in the table are those cells where no rails were found, along with any cells that did not have power or ground nets (in this case there are no such cells).

  The cells share common rails, so they are used to initialize the power rail table option.

```
Abstract: 2001/10/26 08:18:59:

Info:     Power Rail Analysis:

================================================================================
==
Rails found
================================================================================
==
rail  type    net  layer   width  offset  shape     direction   cells
--------------------------------------------------------------------------------
--
1     power   vdd  METAL1  6      46.5    abutment  horizontal  all
2     ground  vss  METAL1  6      -1.5    abutment  horizontal  all

Abstract: 2001/10/26 08:18:59:

Info: Using power rails 1 and 2 to set terminal properties. These settings can
be altered using the Adjust "Power rail widths, offsets and shape"  table.
```

  Example 2: In this scenario, no rail data has been specified by the user in the *Power rail widths, offsets and shape* option table.

  Viewing the table below, it can be established that no common rails were found, therefore, the option table was not initialized. This usually occurs when cells have mismatched rails and should be in different bins.

```
Abstract: 2001/10/26 08:55:27:

Info:     Power Rail Analysis:

================================================================================
==
Rails found
================================================================================
==
rail  type    net  layer   width  offset  shape     direction   cells
--------------------------------------------------------------------------------
```

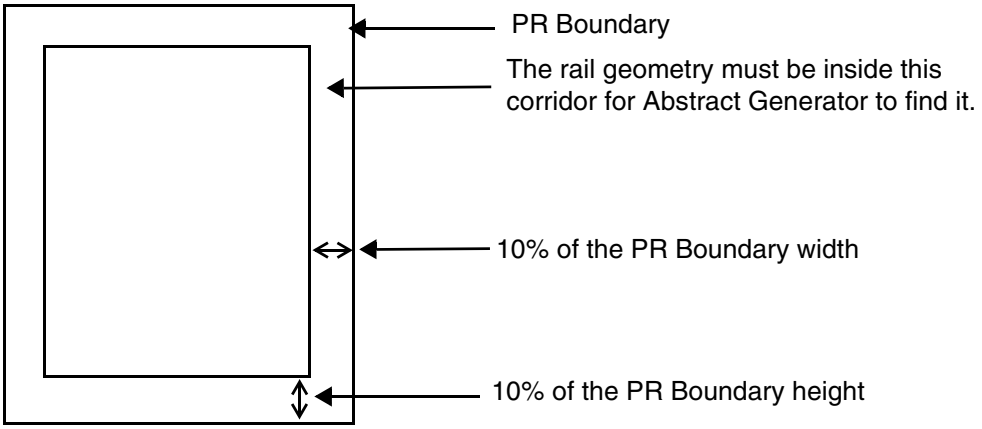

© 2023 Cadence Design Systems, Inc.

```
--
1     power   vdd  METAL1  4      48.5    abutment  horizontal  sc_a2iaia
                                                                sc_a2iaia_10
                                                                sc_a2iaia_11
                                                                sc_a2iaia_12
                                                                sc_a2iaia_2
                                                                sc_a2iaia_3
                                                                sc_a2iaia_4
                                                                sc_a2iaia_5
                                                                sc_a2iaia_6
                                                                sc_a2iaia_7
                                                              more ... total
of 12

2     power   vdd  METAL1  6      46.5    abutment  horizontal  sc_and8b
                                                                sc_and8b_10
                                                                sc_and8b_11
                                                                sc_and8b_12
                                                                sc_and8b_2
                                                                sc_and8b_3
                                                                sc_and8b_4
                                                                sc_and8b_5
                                                                sc_and8b_6
                                                                sc_and8b_7
                                                              more ... total
of 12

3     ground  vss  METAL1  4      -1.5    abutment  horizontal  sc_a2iaia
                                                                sc_a2iaia_10
                                                                sc_a2iaia_11
                                                                sc_a2iaia_12
                                                                sc_a2iaia_2
                                                                sc_a2iaia_3
                                                                sc_a2iaia_4
                                                                sc_a2iaia_5
                                                                sc_a2iaia_6
                                                                sc_a2iaia_7
                                                              more ... total
of 12

4     ground  vss  METAL1  6      -1.5    abutment  horizontal  sc_and8b
                                                                sc_and8b_10
                                                                sc_and8b_11
                                                                sc_and8b_12
                                                                sc_and8b_2
                                                                sc_and8b_3
                                                                sc_and8b_4
                                                                sc_and8b_5
                                                                sc_and8b_6
                                                                sc_and8b_7
                                                              more ... total
of 12

Abstract: 2001/10/26 08:55:27:

Warning: Cannot find power and ground rails common to all cells in the bin.
Specify using the Adjust “Power rail widths, offsets and shape”  table.
```


© 2023 Cadence Design Systems, Inc.

Example 3: In this scenario, the user has not specified any rail data because common rails have not been found in all cells in the bin, the *Power rail widths*, *offsets* and *shape* option table is not initialized.

Cells containing no rails were also found, as were cells that did not contain power and ground nets. The scenario where a cell has no rail identified can arise because the net geometry does not extend far enough out to the edge of the cell. Abstract Generator looks for rail geometry in a corridor within 10% of the dimension of the PR boundary.



Cells that do not contain at least one power and ground net pair are also flagged. This might occur if an incorrect text-to-pin map has been used during the Pins step, or if text labels do not overlap the metal geometry.

```
Abstract: 2001/10/26 11:55:41:

Info:     Power Rail Analysis:

================================================================================
==
Rails found
================================================================================
==
rail  type    net  layer   width  offset  shape     direction   cells
--------------------------------------------------------------------------------
--
1     power   vdd  METAL1  6      46.5    abutment  horizontal  sc_a2iaia
                                                                 sc_a2iaia_10
                                                                 sc_a2iaia_11
                                                                 sc_a2iaia_12
                                                                 sc_a2iaia_13
                                                                 sc_a2iaia_14
                                                                 sc_a2iaia_15
                                                                 sc_a2iaia_16
                                                                 sc_a2iaia_17
                                                                 sc_a2iaia_18
                                                               more ... total
of 25

2     ground  vss  METAL1  6      -1.5    abutment  horizontal  sc_a2iaia
```


© 2023 Cadence Design Systems, Inc.

```
                                                                    sc_a2iaia_10
                                                                    sc_a2iaia_11
                                                                    sc_a2iaia_12
                                                                    sc_a2iaia_13
                                                                    sc_a2iaia_14
                                                                    sc_a2iaia_15
                                                                    sc_a2iaia_16
                                                                    sc_a2iaia_17
                                                                    sc_a2iaia_18
                                                                 more ... total
of 25


No rails found in any of the following cells:
---------------------------------------------
sc_and2a sc_and2a_10 sc_and2a_11 sc_and2a_12
sc_and2a_13 sc_and2a_14 sc_and2a_15 sc_and2a_16
sc_and2a_17 sc_and2a_18 sc_and2a_19 sc_and2a_2
sc_and2a_20 sc_and2a_21 sc_and2a_22 sc_and2a_23
sc_and2a_24 sc_and2a_25 sc_and2a_3 sc_and2a_4
sc_and2a_5 sc_and2a_6 sc_and2a_7 sc_and2a_8 sc_and2a_9


None of the following cells have power and ground nets:
-------------------------------------------------------
sc_syncdflop sc_syncdflop_10 sc_syncdflop_11 sc_syncdflop_12
sc_syncdflop_13 sc_syncdflop_14 sc_syncdflop_15 sc_syncdflop_16
sc_syncdflop_17 sc_syncdflop_18 sc_syncdflop_19 sc_syncdflop_2
sc_syncdflop_20 sc_syncdflop_21 sc_syncdflop_22sc_syncdflop_23
sc_syncdflop_24 sc_syncdflop_25 sc_syncdflop_3 sc_syncdflop_4
sc_syncdflop_5 sc_syncdflop_6 sc_syncdflop_7 sc_syncdflop_8
sc_syncdflop_9
```

- **Values are Specified for the Net Name and Rail Width and Offset**

  Using the same library used in above example, the *Power rail widths, offsets and shape* option table is set with values of (`vdd 6 46.5`) and (`vss 6 -1.5`).

  This time cells are split into two separate tables, depending on whether they contain the user- specified rails.

```
Abstract: 2001/10/26 10:52:47:

Info:     Power Rail Analysis:

================================================================================
==
User specified rails
================================================================================
==
rail  type    net  layer   width  offset  shape     direction   cells
--------------------------------------------------------------------------------
--
1     power   vdd  METAL1  6      46.5    abutment  horizontal  sc_and8b
                                                                 sc_and8b_10
                                                                 sc_and8b_11
                                                                 sc_and8b_12
                                                                 sc_and8b_2
                                                                 sc_and8b_3
                                                                 sc_and8b_4
```


© 2023 Cadence Design Systems, Inc.

```
                                                                    sc_and8b_5
                                                                    sc_and8b_6
                                                                    sc_and8b_7
                                                                 more ... total
of 12

2     ground  vss  METAL1  6      -1.5    abutment  horizontal  sc_and8b
                                                                    sc_and8b_10
                                                                    sc_and8b_11
                                                                    sc_and8b_2
                                                                    sc_and8b_3
                                                                    sc_and8b_4
                                                                    sc_and8b_5
                                                                    sc_and8b_6
                                                                    sc_and8b_7
                                                                 more ... total
of 12

================================================================================
==
Rails found in cells not containing user specified rails
================================================================================
==
rail  type    net  layer   width  offset  shape     direction   cells
--------------------------------------------------------------------------------
--
1     power   vdd  METAL1  4      48.5    abutment  horizontal  sc_a2iaia
                                                                    sc_a2iaia_10
                                                                    sc_a2iaia_11
                                                                    sc_a2iaia_12
                                                                    sc_a2iaia_2
                                                                    sc_a2iaia_3
                                                                    sc_a2iaia_4
                                                                    sc_a2iaia_5
                                                                    sc_a2iaia_6
                                                                    sc_a2iaia_7
                                                                 more ... total
of 12

2     ground  vss  METAL1  4      -1.5    abutment  horizontal  sc_a2iaia
                                                                    sc_a2iaia_10
                                                                    sc_a2iaia_11
                                                                    sc_a2iaia_12
                                                                    sc_a2iaia_2
                                                                    sc_a2iaia_3
                                                                    sc_a2iaia_4
                                                                    sc_a2iaia_5
                                                                    sc_a2iaia_6
                                                                    sc_a2iaia_7
                                                                 more ... total
of 12
```

- **The Rail Data Exceeds the Character Limit**

  In this scenario, the data selection exceeds the size limit of characters defined by SKILL for the Power Rail Analysis report.

  The Power Rails Analysis report is divided such that each block of message can contain only a limited number of characters in a line. For example, if 100 rail entries are retrieved

© 2023 Cadence Design Systems, Inc.

and the threshold is 80, the first message `INFO (ABS-14004)` prints the data of the first 80 rails and the remaining data of the next 20 rails is displayed using the `INFO (ABS-19048)` message.

```
INFO (ABS-14004): Power Rail Analysis:
============================================================
Rails found
============================================================
rail  type    net         layer  width  offset   shape     direction   cells
------------------------------------------------------------------------------
------
1     ground  gnd         M0     0.06   5.25     abutment  horizontal  cell_1
2     ground  gnd         M0     0.06   5.73     abutment  horizontal  cell_1
3     ground  gnd         M0     0.06   6.21     abutment  horizontal  cell_1
4     ground  gnd         M0     0.06   6.69     abutment  horizontal  cell_1
5     ground  gnd         M0     0.06   0.93     abutment  horizontal  cell_1
.
.
.
79    ground  gnd         M5     0.076  5.396    abutment  vertical    cell_1
80    ground  gnd         M5     0.076  53.96    abutment  vertical    cell_1

INFO (ABS-19048): continued...

81    ground  gnd         M6     0.076  54.582   abutment  horizontal  cell_1
82    ground  gnd        M5     0.076  54.644   abutment  vertical      cell_1
83    ground  gnd        M5     0.076  55.328   abutment  vertical      cell_1
84    ground  gnd         M6     0.076  55.542   abutment  horizontal  cell_1
84    ground  gnd         M6     0.076  55.542   abutment  horizontal  cell_1
85    ground  gnd        M5     0.076  56.012   abutment  vertical      cell_1
.
.
.
99    ground  gnd         M5     0.076  6.08     abutment  vertical    cell_1
100   ground  gnd        M5     0.076  60.8     abutment  vertical    cell_1
```

Related Topics

Specifying Power and Ground Rail Characteristics

© 2023 Cadence Design Systems, Inc.

Abstracts for Routed Blocks with Power Rings

There might be situations in which the cell data imported comes from a DEF file. This is usually a routed block for which you might want to create an abstract to use at a higher level of a design.

Abstract Generator can generate abstracts from routed blocks. This includes the ability to create abstracts for blocks with ring power pins, for blocks with cover blockages inside the power ring, and for blocks with detailed obstructions for signal nets crossing the power rings.

Power Pins

Consider the following points when deciding which approach to use when dealing with power ring pins:

1. Before you decide whether or not power extraction is necessary.

   Power extraction is not required after running the *Pins* step, if the power pins contain at least the power pin geometry. They will contain power pin geometry if the power ring shapes were defined as physical pins in the imported DEF file. This is unlikely, and in the case where no physical pins were defined in the DEF file, the Pins step will create pins from all of the power routing in the block (including the power ring).

   If extraction is required, perform a limited power extraction by limiting the hierarchy depth. This will result in the extraction of the power ring plus some other top level shapes. As a recommendation, set extraction depth to `1` so that Abstract Generator finds the ring pins plus the via geometry. Go to step 2.

   If there is no need to extract, leave power extraction turned off and consider the Abstract step requirements in step 2.

2. In the *Abstract* step, ensure that the *Create ring pins* option is turned on. The algorithm to determine if a power pin should be preserved as a ring pin is based on *distance to boundary*. The closest power pin to the boundary is selected along with any others that are equidistant. Therefore, if a power ring is equidistant to the boundary on all sides, then everything should be fine.

   Otherwise, you will have to manually control the settings in the *Ring pin max distance to boundary* field, setting it to be equal to the maximum distance from the boundary within which you want any power shapes to be considered as ring pins.

   You must set this based on the distance to the *inner ring* (VDD or VSS, whichever is inner).

© 2023 Cadence Design Systems, Inc.

Signal Pins

If the signal pins read in from the DEF file are sufficient, there is no need to perform connectivity extraction, and no requirement to have boundary pins turned on. Therefore, turn he *Create boundary pins* option off since it is on by default in the *Block* bin.

Alternatively, if your signal pins are insufficient, you would run connectivity extraction and leave the *Create boundary pins* option turned on.

Obstructions

To form the necessary obstructions with the region inside the power ring as a solid obstruction and with the signals that cross the ring into obstructions in that region, set the Blockage option to *Cover* in the *Layer Assignment for Blockages* table and specify the cover blockage distance to boundary value.

Related Topics

Running step Abstract Form

Customizing Pin Shapes in Standalone Abstract Generator

© 2023 Cadence Design Systems, Inc.

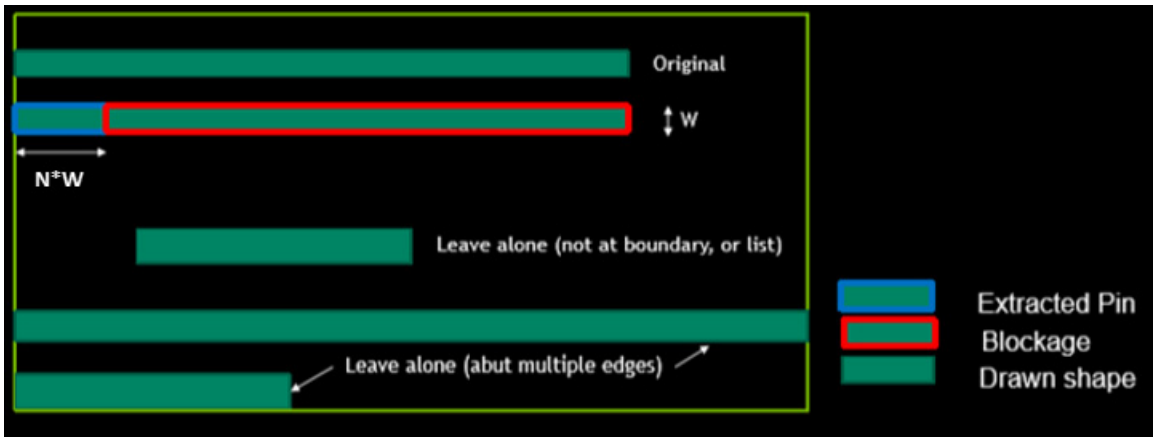
Boundary Pin Trimming

Abstract Generator lets you trim long pins that touch the PR boundary in the abstract view and extend deep inside the cell. This topic describes how to trim boundary pins by any factor.

Set the `AbstractAdjustTrimPins` option to `true` in `absAbstract` SKILL function to enable this functionality. Abstract Generator processes only those pins that touch a single edge of the PR boundary. Pins that touch multiple edges of the PR boundary, or pins that straddle the PR boundary are ignored for processing.

The trim factor is an integral multiple of the width of the pin. You can provide the trim factor by specifying it using the `AbstractAdjustTrimPinsFactor` option. After trimming is complete, the remaining section of the pin is converted into a detailed blockage.

In the image below, `W` specifies the width of the pin and `N` specifies the trim factor.



To exclude certain nets, specify the net names using regular expressions in the `AbstractAdjustExcludeNetNames` option.

Use the following syntax to specify the options in the abstract options file:

- `absSetBinOption("Core" "AbstractAdjustTrimPins" "true")`
- `absSetBinOption("Core" "AbstractAdjustTrimPinsFactor" "4")`
- `absSetBinOption("Core" "AbstractAdjustExcludeNetNames" "^((V(DD|CC))|(v(dd|cc)))(!)?$")`

Note: This functionality is supported for all rectilinear shapes.

© 2023 Cadence Design Systems, Inc.

Related Topics

absExtract

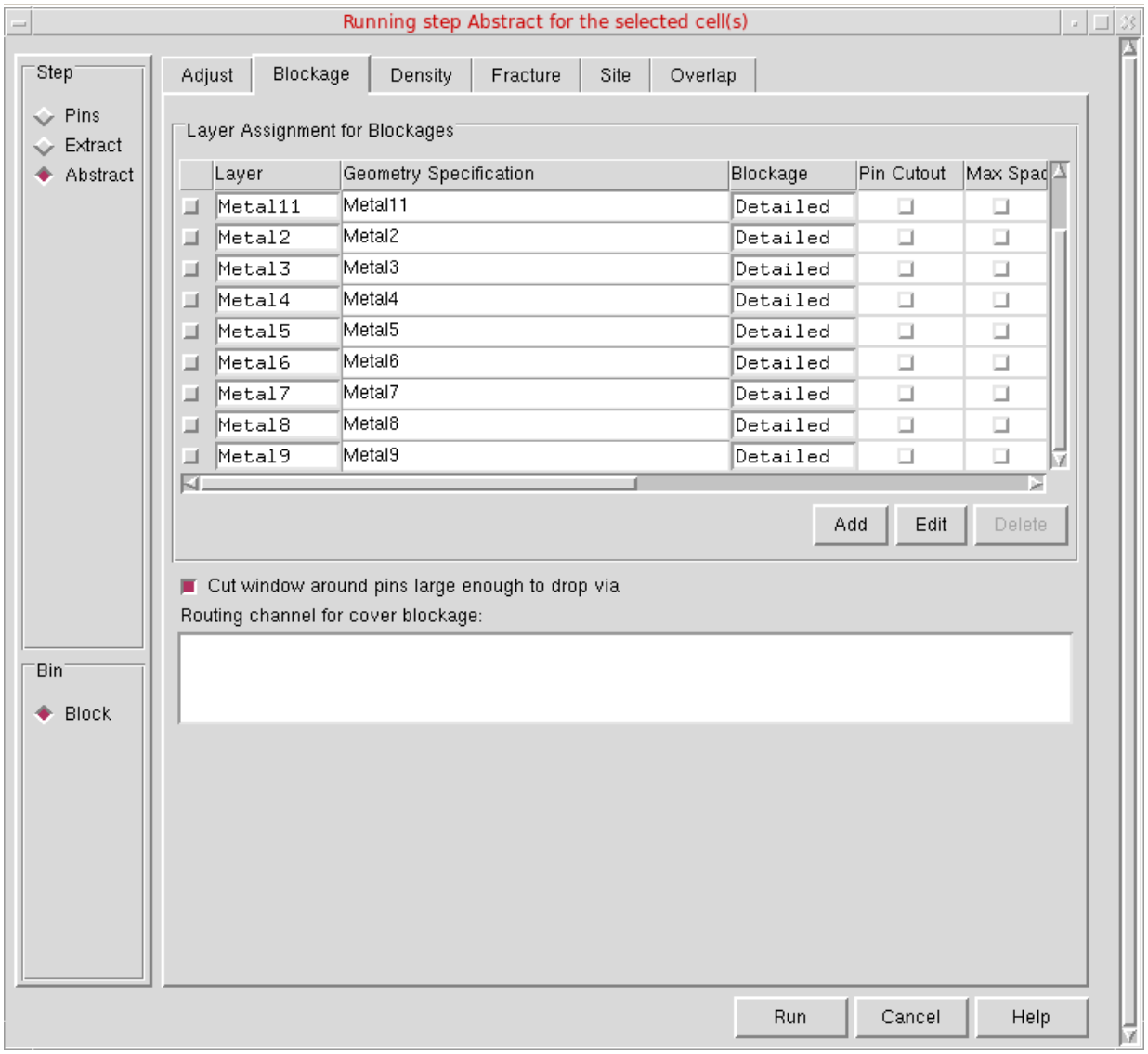
Customizing Pin Shapes in Standalone Abstract Generator

© 2023 Cadence Design Systems, Inc.

Modifying Blockage Geometry in the Abstract View

You can modify the blockage geometries in the abstract view using the *Blockage* tab in the Running step Abstract form.

1. Choose *Flow – Abstract* and click the *Blockage* tab in the Running step Abstract form.



   The options on this tab control the final blockage geometry in the abstract view. Any blockage geometry already existing in the view is preserved and added to the final calculated blockages.

2. Use the *Layer Assignment for Blockages* table to specify how blockages are to be modeled in the abstract.

© 2023 Cadence Design Systems, Inc.

3. Select *Cut window around pins large enough to drop via* to control the size of the window cut into a blockage around a pin.
4. Use *Routing channel for cover blockage* to create a routing channel when generating a cover blockage.

   (Metal1 (y9 drawing)) (Metal2 (y9 drawing))

5. Click *Run*.

Related Topics

Running step Abstract Form

© 2023 Cadence Design Systems, Inc.

Performance Improvements for Generating Detailed Blockages

If you are creating *Detailed* blockages for layouts that have big blockages at the top level, you can gain considerable performance benefits by the use model described here.

This use model is implemented only if the geometry specification (see the criteria below) for a blockage layer is such that the top-level shapes that satisfy the geometry specification are not excluded when the same geometry specification is applied at all the levels. In such a situation, the shapes are processed in the following way:

- At the top level, shapes satisfying the geometry specification and that are bigger than 10 percent of the total cell area are considered as “big blockages”.
- At the lower levels, shapes on the boundary purpose and that are bigger than 10 percent of the total cell area are considered as “big blockages”.

After the “big blockages” are identified, they are processed in the same way as *Cover* blockages. This use model greatly reduces the complexities of processing and merging shapes and also reduces the number of shapes that are ultimately processed for blockage creation.

Use the `BlockageLargeShape` option to enable Abstract Generator to look for big blockages in the current design. The `BlockageLargeShape` option is always used in association with the `BlockageLargeShapePct` option. The size of the identified blockages is increased by the percentage specified by the `BlockageLargeShapePct` option, and a cover blockage is created.

Use the `BlockageLargePurposeList` option to specify the purpose of the identified big blockages.

Use the following criteria for writing a valid geometry specification:

- Geometry specifications with a single purpose:
  - The boundary purpose should be specified explicitly. For example,

    ```
    M1 boundary
    ```

  - If any purpose other than the boundary purpose is specified, there should be no value specified in the *Ignored Purpose(s)* field (*File – General Options – General* tab). For example,

    ```
    M1 net
    M1 drawing
    ```

- Geometry specifications with more than one purpose:

© 2023 Cadence Design Systems, Inc.

- The boundary purpose should be specified along with the other purpose(s) by using only the OR operator. For example,

  ```
  M1 boundary or M1 drawing
  ```

- If any combination of LPPs excluding the boundary purpose is specified, there should be no value specified in the *Ignored Purpose(s)* field (*File – General options – General* tab). For example,

  ```
  M1 net or M1 drawing
  ```

Related Topics

Running step Abstract Form

Modifying Blockage Geometry in the Abstract View

© 2023 Cadence Design Systems, Inc.

Control Spacing Value During Geometry Add Backs

When Abstract Generator creates a cover blockage with pin cutout, it automatically adds back geometries that are located near the boundaries. Geometries that are located between the metal_max_spacing value and metal_min_spacing value are added back. Depending on your requirement, you can specify the distance within which the geometries need to be added back. For example, for some designs, a large number of geometries may be located within this distance. In such cases, instead of adding back all the geometries, you can reduce the spacing value. Use the `absSetBinOption( "Core" "AbstractBlockageUserDefinedDistanceForAddBack" "")` option to control the spacing values.

Example

```
absSetBinOption( "Core" "AbstractBlockageUserDefinedDistanceForAddBack" "(M1 0.5) (M2 0.6)")
```

In the above example, for M1 cover blockage, add back geometries will be searched within distance of 0.5 and for M2 cover blockage, add back geometries will be searched within the distance of 0.6.

In the cover blockage model, all shapes of the specified layer that are close to or are partially overlapping with the PR boundary are retained, and created as addbacks. Shapes of the specified layer that are deep inside the PR boundary or are completely outside the PR boundary are lost.

In the detailed blockage model, all shapes of the specified layer that are inside, outside, or overlapping the PR boundary are created as blockages.

Related Topics

Running step Abstract Form

Modifying Blockage Geometry in the Abstract View

© 2023 Cadence Design Systems, Inc.

Impact of Cutout and Spacing Options on Blockage Types

This topic describes how each of the three types of blockages behave with the *Pin Cutout*, *Max Space*, and *Corridor Cut* options.

Cover Blockage

A *Cover* blockage covers the entire block. The default *Cover* blockage does not have cutouts either around pins or at the block boundary and is tagged with the spacing attribute `+SPACING` set to `maxSpacing`. Therefore, the top-level routes stay `maxSpacing` away from the block. The *Cover* blockage in this state offers the least cross coupling between the routes at the top level and those within the block. The top-level routes can access pins at the boundary and the pins within the block can be accessed from the top in a perpendicular direction.

For a default *Cover* blockage generation, neither of *Pin Cutout*, *Corridor Cut*, and *Max Space* option is selected.

- *Pin Cutout* is OFF

  Abstract Generator will not create cutouts around pins. A large blockage is added that will cover the entire block.

- *Pin Cutout* is ON

  Abstract Generator creates pin cutouts at `minSpacing` distance away from the pins. If the layer-purpose-based and voltage dependent rules are present in the technology file along with layer based spacing rules and option `RespectPurposeAndVoltageRules`, to conform to these rules, is set to TRUE, then Abstract Generator will read the layer-purpose rules, voltage-based rules, and the layer rules. A blockage around the pin is created with the maximum value of the `minSpacing` available based on the layer-purpose-based and voltage dependent rules. The use of the *Pin Cutout* option is recommended for generating a *Cover* blockage to obtain a densely packaged block.

- *Pin Cutout* and *Corridor Cut* are ON

  Abstract Generator creates corridor cuts for internal pins in addition to creating the pin cutouts. The creation of corridor cuts is subject to the existence of straight-line paths from internal pins to the block boundary (for a `minWidth` wire). The *Pin Cutout* option should be selected for selecting the *Corridor Cut* option.

- *Pin Cutout* and *Max Space* are ON

  In addition to the pin cutouts, Abstract Generator creates a blockage at the block boundary that keeps the top-level routes at `maxSpacing` away from the block. This

© 2023 Cadence Design Systems, Inc.

further reduces the cross coupling between internal and top-level routes. The *Max Space* option is functional only when the *Pin Cutout* option is selected. The *Corridor Cut* option is ineffective when both these options are selected.

Shrink Blockage

A *Shrink* blockage covers the small, less useful areas in a block. The default *Shrink* blockage does not have cutouts either around pins or at the block boundary.

In the case of *Shrink* blockages, no blockage is added around the block at the boundary. Therefore, the *Max Space* option is ineffective in the case of *Shrink* blockages even if selected.

- *Pin Cutout* is ON

  Abstract Generator creates pin cutouts at `minSpacing`.

- *Pin Cutout* and *Corridor Cut* are ON

  Abstract Generator creates corridor cuts for internal pins. The creation of corridor cuts is subject to the existence of paths from internal pins to the block boundary (for a `minWidth` wire). The *Pin Cutout* option should be selected for the *Corridor Cut* option to be effective.

Detailed Blockage

A *Detailed* blockage covers geometries that are not part of pins on the routing layers. The default *Detailed* blockage does not have cutouts either around pins or at the block boundary.

In the case of *Detailed* blockages, no blockage is added around the block at the boundary and no corridors are cut around pins. Therefore, both *Max Space* and *Corridor Cut* options are ineffective in the case of *Detailed* blockages even if selected.

- *Pin Cutout* is OFF

  Abstract Generator does not create cutouts around pins.

- *Pin Cutout* is ON

  Abstract Generator first creates pin cutouts on the *Detailed* blockage. These pin cutouts are computed using the appropriate spacing rules, which in turn, are based on the respective pin's width. Next, Abstract Generator determines instances of pin cutouts in which the cutout width is greater than the `minSpacing` for the layer. For these instances, Abstract Generator adds small blockages between the `minSpacing` and the pin cutouts.

© 2023 Cadence Design Systems, Inc.

Finally, the small added blockages are tagged with the spacing attribute `+SPACING` set to `minSpacing`.

Note: The additional blockages are added to prevent top-level routes from entering the pin cutout area.

The following table summarizes the effect of selecting various combinations of these options for the three types of blockages. The red button indicates that the option is selected and the white button indicates that the option is not selected. The symbol 'X' for an option indicates the ineffectiveness of the option in that particular scenario even if selected.

Blockage Type	Options in the Table on the Blockage Tab			Resultant Blockage
	Pin Cutout	Max Space	Corridor Cut	
Cover	☐	X	X	Default cover blockage: - No pin cutouts - No corridor cuts - No blockage at the boundary
	■	☐	☐	Cover blockage with pin cutouts. No corridor cuts.
	■	☐	■	Cover blockage with pin cutouts and corridor cuts for internal pins.
	■	■	X	Cover blockage with pin cutouts and blockage at boundary to keep routers `maxSpacing` away. No corridors cuts.
Shrink	☐	X	X	Default shrink blockage: - No pin cutouts - No corridor cuts - No blockage at the boundary
	■	X	☐	Shrink blockage with pin cutouts. No corridor cuts.
	■	X	■	Shrink blockage with pin cutouts and corridor cuts for internal pins.

© 2023 Cadence Design Systems, Inc.

Blockage Type	Options in the Table on the Blockage Tab			Resultant Blockage
	Pin Cutout	Max Space	Corridor Cut	
Detailed	☐	X	X	Default detailed blockage: - No pin cutouts - No corridor cuts - No blockage at the boundary
	■	X	X	Detailed blockage with pin cutouts. No corridor cuts.

■ Selected
☐ Not Selected
X Not Applicable

Related Topics

Running step Abstract Form

Modifying Blockage Geometry in the Abstract View

© 2023 Cadence Design Systems, Inc.

Pin Cutout Support on Metal Layers Above the Pin Layer

When the Pin Cutout option is enabled, Abstract Generator creates cutouts on the layer that contains pins. To create pin cutouts on blockage layers above the layers that contains pins, you can use the `AbstractBlockageCutForAboveLayers` option available in the Abstract step of Abstract Generator. Specifying the `AbstractBlockageCutForAboveLayers` option enables the router to connect to pins by using stacked vias. This option is not available in the GUI.

The `AbstractBlockageCutForAboveLayers` takes precedence over all other cut distance related options such as `AbstractBlockageDownPinCutWindow` and `BlockageCutVia`.

With the `AbstractBlockageCutForAboveLayers` option, you specify a valid metal layer that contains pins and the size of the pin cutouts to be created in the blockage layers above this layer. Specifying the size of the pin cutout is al. If the pin cutout size is not specified, the size of the pin cutout is determined by the minspacing of the blockage layer. The following example illustrates this:

```
absSetBinOption(“Block” “AbstractBlockageCutForAboveLayers” “(Metal2 0.1) (Metal4 0.2)”)
```

Here, for pins on `Metal2` layer, the blockages on `Metal3`, `Metal4`,…,`MetalX` are cut with a distance of `0.1u;` for pins on `Metal4` layer, the blockages on `Metal5`, `Metal6`,…,`MetalX` are cut with a distance of 0.`2u`.

The following is another example that specifies pin cutouts:

```
absSetBinOption(“Block” “AbstractBlockageCutForAboveLayers” “Metal2 (Metal4 0.2)”)
```

Here, for pins on `Metal4` layer, the blockages on `Metal5`, `Metal6`,…,`MetalX` are cut with a distance of `0.2u`. For pins on `Metal2` layer, the blockages are cut with a minimum spacing specified for the layer on which the blockage is created. This is because the pin cutout value is not specified for `Metal2` layer. Therefore, the blockages on `Metal3` are cut with `Metal3` minSpacing and blockages on `Metal4` are cut with `Metal4` minSpacing, and so on.

The `AbstractBlockageCutForAboveLayers` option works only for those layers for which *Pin Cutout* is enabled. In the example below, pin cutout is enabled only for `Metal2` to `Metal5` layers. Therefore, blockages will be cut only for these layers.

```
absSetBinOption("Block" "AbstractBlockageCutAroundPin" "Metal2 Metal3 Metal4 Metal5")
```

Related Topics

Modifying Blockage Geometry in the Abstract View

© 2023 Cadence Design Systems, Inc.

Routing Channel Creation when Generating Cover Blockages

Abstract generator enables you to create routing channels through cover blockages. Consider layer `Metal1`. When generating a cover obstruction on this layer, you can choose to cut a routing channel through this layer. The routing channel will take the shape of the layer-purpose pair, which you specify as a parameter. In this example, the routing channel will be in accordance with (y9, drawing) layer-purpose pair.

Note: This feature is not available for shrink and detailed blockages.

To create a routing channel when generating a cover blockage, specify the parameters in the *Routing channel for cover blockage* field.

Here, *Metal1* refers to layer through which a routing obstruction needs to be cut. *(y9 drawing)* is the layer-purpose pair that indicates the shape of the routing channel.

You can use the `AbstractBlockageRoutingChannelOnLayers` option to create a routing channel when generating a cover blockage.

Example

```
absSetBinOption("Block" "AbstractBlockageRoutingChannelOnLayers" "(Metal1 (y9
drawing)) (Metal2 (y9 drawing))")
```


© 2023 Cadence Design Systems, Inc.

Layers `(y [0-9] drawing)` are system-defined layers. So, the results may not be accurate when these layers are used for creating routing channels.

Related Topics

Modifying Blockage Geometry in the Abstract View

© 2023 Cadence Design Systems, Inc.

Automatic Pin Stretching

Important

The following section is only for information. There is no option to enable “automatic pin stretching”.

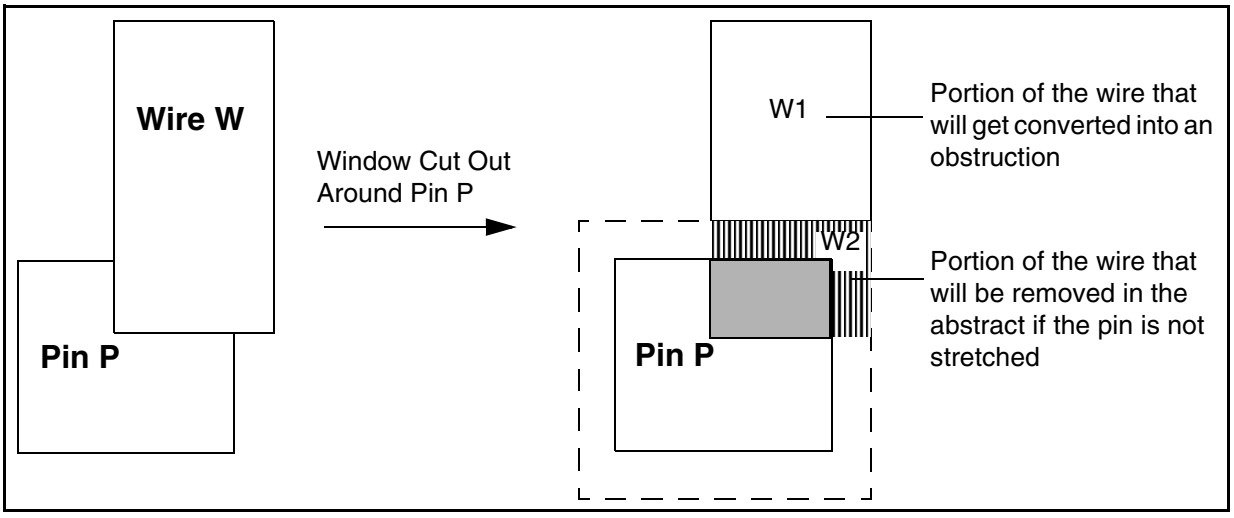
Abstract Generator sometimes requires to make the pin geometry larger than the settings you have specified. This feature is known as *automatic pin stretching* and is used in circumstances where cutting a window around a pin would remove some geometry (obstruction) that touches the pin, potentially letting the router route too close to the cut-out geometry. See the example below.

Automatic pin stretching searches for geometry only to the *Maximum depth* of hierarchy as specified in the Extract step’s *Signal* or *Power* tab. The default depth is 32.

Example

This is an example of a situation in which pin stretching occurs. The example also illustrates the reasons why pin stretching occurs.

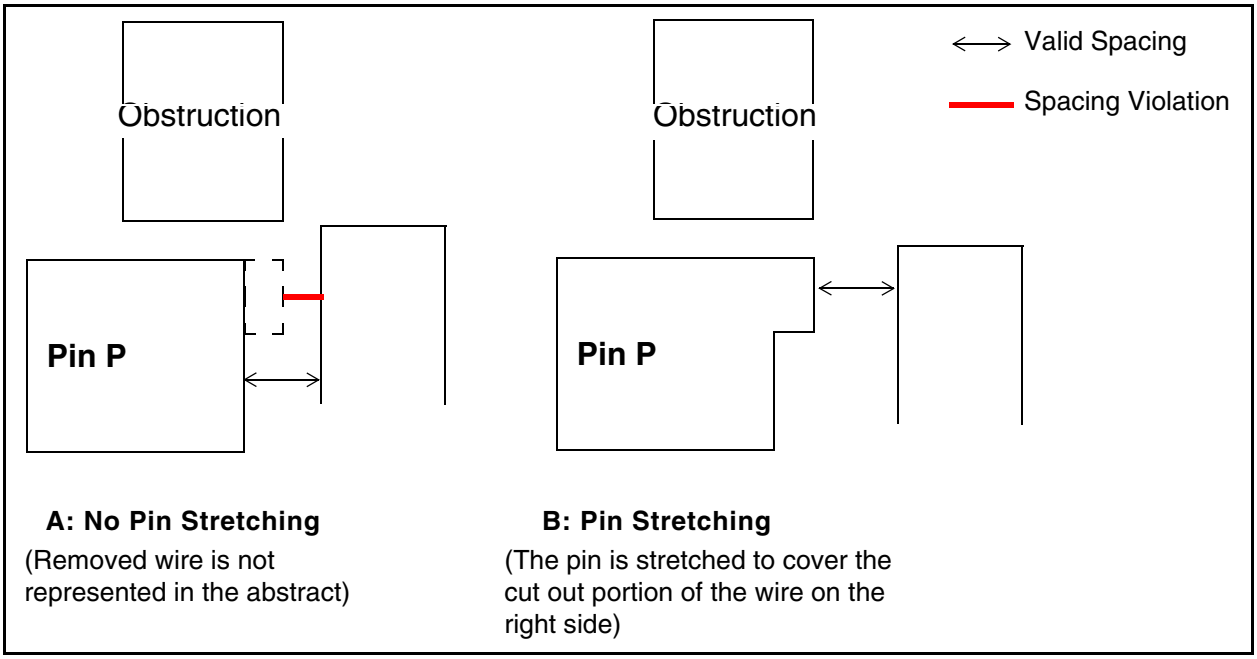
The following figure shows the wire geometry W touching the Pin P. When a window is cut out around Pin P, the wire geometry that touches the pin at the right edge, W2, is totally removed in the abstract. W1 part of the wire geometry becomes an obstruction. The W2 wire geometry cannot also be converted into an obstruction because it covers the area in which a window has already been cut out around Pin P.



The figure below shows both the outcomes when Pin P is either stretched (**B**) or not stretched (**A**) towards the right side. In **A**, when the pin is not stretched, the removed wire geometry is

© 2023 Cadence Design Systems, Inc.

not represented in the abstract. So it is possible for the router to route a wire within the minimum separation distance to the right side of Pin P. This can lead to spacing violation with the W2 portion of the original wire when the abstract is replaced with the pin and wire geometries in the layout.



Therefore, the W2 wire geometry must be represented in the abstract somehow. In this situation, the pin is automatically stretched to the right to cover the portion of the wire that cannot be turned into an obstruction (W2). Since the resultant stretched pin includes the wire geometry, the router will now space away a new wire away appropriately from the W2 wire geometry. This is shown in **B** in the figure above.

There is no need to extend Pin P vertically above because even when the window is cut out around the pin, there is still a piece of the wire (W1) that is converted into an obstruction above Pin P. Therefore, unless the cut out around the pin is large, the space between Pin P and the new obstruction will not be large enough for the router to route unrelated wiring through.

By default, pin stretching occurs automatically whenever a window cut out around a pin removes some geometry touching the pin. If you want, you can disable pin stretching by setting the *Cut Same* value to 0 and deselecting the *Cut window around pins large enough to drop via* option.

Related Topics

Non-Square Boundary Pins

© 2023 Cadence Design Systems, Inc.

Modifying Blockage Geometry in the Abstract View

© 2023 Cadence Design Systems, Inc.

Calculating Metal Layer Density

You can generate metal density information for abstracts in LEF 5.8. The density value in LEF helps in verifying metal density fabrication rules early in the design cycle without having to replace the abstracts with the layouts. This information in LEF can reduce the memory requirements and increase the speed of metal fill tools and aid these tools to analyze and fix the designs using abstracts.

To calculate the metal layer density:

1. Choose *Flow – Abstract* and click the *Density* tab in the Running step Abstract for the selected cell(s) form.

© 2023 Cadence Design Systems, Inc.

The options on this tab provide specifications to Abstract Generator for calculating metal density (in percentage) for macros.

2. Select *Calculate metal density* to specify that metal density information is to be generated for the selected cell(s).

   Now, you can specify the details to enable Abstract Generator in determining the metal layers and metal geometries for which density calculation needs to be performed.

3. You can use the same metal layers and their corresponding geometry specifications as you would have specified for signal nets, power nets, or antenna data extraction in the Extract step. To do this, select one of the following options on the *Density* tab:

   - *Use layer assignment for signal extraction*
   - *Use layer assignment for antenna extraction*
   - *Use layer assignment for power extraction*

4. You can specify a different set of geometry specifications for metal layers by using the *Layer Assignment for Metal Density Regions* table. To use the geometry specifications from this table, ensure that none of the three layer assignment options is selected.

5. Use the following fields to specify the same width and height value for all the layers:

   - *Default density window width*
   - *Default density window height*

6. Click *Run*.

Related Topics

Density Calculation

Running step Abstract Form

© 2023 Cadence Design Systems, Inc.

Density Calculation

You can generate metal density information for abstracts in LEF 5.8. The density value in LEF helps in verifying metal density fabrication rules early in the design cycle without having to replace the abstracts with the layouts. This information in LEF can reduce the memory requirements and increase the speed of metal fill tools and aid these tools to analyze and fix the designs using abstracts.

Specifying the Metal Layers for Density Calculation

For metal density calculation, you can either use the already specified metal layers and their geometry specifications or specify them afresh on this tab. These two alternatives are elaborated below:

- You can use the same metal layers and their corresponding geometry specifications as you would have specified for signal nets, power nets, or antenna data extraction in the Extract step. To do this, select one of the following options on the *Density* tab:
  - *Use layer assignment for signal extraction*
  - *Use layer assignment for antenna extraction*
  - *Use layer assignment for power extraction*

  These options are mutually exclusive and selecting one disables the other two options. Only metal layers specified in the selected table will be considered for metal density calculation.

  Along with selecting one of these options, you can ally specify the *Width* and *Height* of the density window in the *Layer Assignment for Metal Density Regions* table. See Specifying the Dimensions of the Density Window.
- Alternatively, you can specify a different set of geometry specifications for metal layers by using the *Layer Assignment for Metal Density Regions* table. To use the geometry specifications from this table, ensure that none of the three layer assignment options is selected. You can turn off these options individually.

  The table contains the following columns:
  - The *Layer* column is pre-populated with the metal layers from the technology file. You can remove or edit the existing layers or add any removed layers. Density data will not be calculated for metal layers that are not specified in the table.

    If you use the *Layer Assignment for Metal Density Regions* table and you have specified a non-metal layer in the table, Abstract Generator will issue a warning message during density calculation.

© 2023 Cadence Design Systems, Inc.

- By default, the *Geometry Specification* column contains the layer names corresponding to each metal layer name in the *Layer* column. You can modify these values as required.
- The *Width* and *Height* columns are described in Specifying the Dimensions of the Density Window section.

Important

If you select to use a layer assignment table for density calculations from a different tab, you can still use the *Width* and *Height* columns of the *Layer Assignment for Metal Density Regions* table on the *Density* tab. In this case, though the geometry specifications in the *Layer Assignment for Metal Density Regions* table will not be considered for metal density calculation but you need to ensure that a valid geometry specification exists for every layer specified in this table.

Specifying the Dimensions of the Density Window

You can specify the width and height of the density window either for all the layers or separately for each layer.

To specify the width and height on a per layer basis, use the *Width* and *Height* columns of the *Layer Assignment for Metal Density Regions* table. You can use the *Width* and *Height* columns of this table when you either select one of the layer assignment options or use the *Layer Assignment for Metal Density Regions* table for density calculation.

To specify the same width and height value for all the layers, use the following fields on the *Density* tab:

- *Default density window width*
- *Default density window height*

The default value for both the fields is `20`, which is derived by calculating 10% of a typical `130nm` window and 20% of a typical `90nm` density check window size. You can modify this value to get Abstract Generator to override the default window size without having to change the *Width* and *Height* values for every layer.

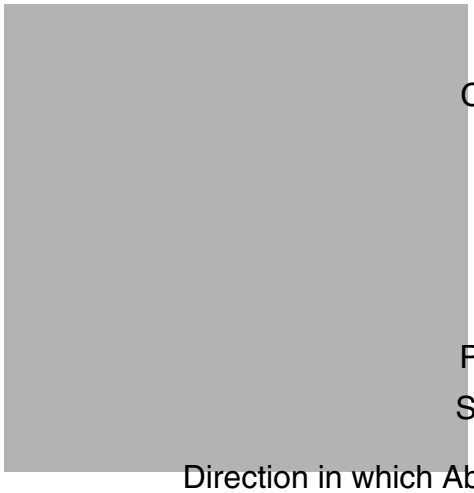
Note: If you do not populate the *Width* or the *Height* column, Abstract Generator picks the density window dimension from the *Default density window width* and *Default density window height* fields.

Some density windows in the final abstract may be smaller in dimension than the option values specified. This happens around the edges of the PR boundary on various layers. This is illustrated in the following scenarios:

© 2023 Cadence Design Systems, Inc.

Scenario I: Rectangular PR Boundary

Starting from the lower left corner of the bounding box of the PR boundary, the abstract is divided into a rectangular grid with window size as specified in the GUI. The rectangles intersecting the PR boundary are clipped at the right and top sides. Therefore, the size of the density windows at the right and top sides will be smaller than the size specified for the density window in the GUI. This is shown in the figure below.



Scenario II: Rectilinear PR Boundary

The density windows are clipped wherever they intersect the PR boundary. The resulting polygons are fractured into smaller rectangles and density data is generated for each of these independently.

Abstract with Rectilinear PR Boundary

Enlarged View of Fractured Density Window

© 2023 Cadence Design Systems, Inc.

Storage Location of Metal Density Information

On OpenAccess, the density rectangles are saved as an `oaLayerBlockage` object with type `oacFillBlockageType`. The bBox of the `oaLayerBlockage` object is reported as the metal density rectangle in LEF by the export tool.

Related Topics

Calculating Metal Layer Density

© 2023 Cadence Design Systems, Inc.

Fracturing Pins and Blockages

You can control whether or not pins and blockages are fractured depending on the PR system you are using. By using the options on the *Fracture* tab in the Running step Abstract form, you can also control the modeling of 45-degree geometry. These options are available to all bins.

To fracture pins and blockages:

1. Choose *Flow – Abstract*, and click the *Fracture* tab in the Running step Abstract form.

2. In the *Fracture* section, you can specify whether you want to fracture pins or blockages by using the options - *Fracture pins* and *Fracture blockages*. By default, both these options are disabled.
3. In the *45-Degree Geometry* section, you can convert a 45-degree geometry.
   - Select *Stair-step coverage* to control the degree to which stair-step blockages cover the 45-degree geometry.
   - Use *Stair-steps width* to specify the width of the blockage geometry to be created.
4. Click *Run*.

© 2023 Cadence Design Systems, Inc.

Related Topics

Density

Running step Abstract Form

Pin and Blockage Fracturing

© 2023 Cadence Design Systems, Inc.

Pin and Blockage Fracturing

LEF 5.8 supports the routing of 45-degree geometries. Abstract Generator provides support for preserving the 45-degree geometries as it is. By default, the conversion of 45-degree geometries to rectilinear ones is disabled. The options to convert a 45-degree geometry into stair-stepped rectangles are available only if one or both of *Fracture pins* and *Fracture blockages* options are active.

Setting the Type of Stair Step Coverage

The *Stair-step coverage* option controls the degree to which stair-step blockages cover the 45-degree geometry. There are three settings available for this option: *full*, *truncate*, and *partial*.

- With full stair-step coverage, the 45-degree geometry is completely obscured by blockage rectangles as shown below:

  This ensures that no real violations are created.

© 2023 Cadence Design Systems, Inc.

- With truncate stair-step coverage, the blockage rectangles are fully enclosed by the original 45-degree geometry as shown below.

  You might use this setting when there is another geometry close enough to cause violations with any stair-steps that extend outside the 45-degree edges.

  Note: There is a considerable risk of the router being able to create routes that would not violate the abstract, but would violate the real 45-degree geometry.

- With partial stair-step coverage, the blockage rectangles only partially cover the 45-degree geometry as shown below.

  This frees up more routing resources for the PR tools but because some of the real 45-degree geometry is exposed, there is an increased risk of violations.

  In some situations, the stair-stepping procedure can create rectangles which introduce violations with neighboring geometry. This effect can be minimized by using a smaller step size or by using *truncate* rather than *partial* or *full* settings. Additional problems can be introduced by using the *truncate* setting. The 45-degree geometry might be inadequately covered, and violations between wiring and 45-degree geometry would go undetected because of the modeling in the abstract. Again, using smaller stair-step distance can minimize this.

Setting the Width of the Stair Step

In the *Stair-steps width* field, you can specify the width of the blockage geometry to be created. Enter a value greater than zero. There is no minimum width restriction. The default value is half the `minSpacing` of the `Metal1` layer. If the `minSpacing` of `Metal1` layer is not found in the technology data, then the default value is set to `5`.

© 2023 Cadence Design Systems, Inc.

Related Topics

Density

Running step Abstract Form

Pin and Blockage Fracturing

© 2023 Cadence Design Systems, Inc.

Specifying Sites in Standalone Abstract Generator

Sites tell the placer where it can place cells. All cells located in a particular bin share a common set of properties: symmetry, class (for example, pad or core), width, and height. These properties are used by the placer to assist in the placement process.

These cell properties are defined in a site object, and all cells within the same bin are associated with the same site. The site itself has a name, and the cells in the LEF file contain a site name field that holds the name of the associated site. You can create sites for the *Core, IO, Corner* bins.

In addition to the site names defined in the technology file, for CORE cells, the <AUTO> option is available in the *Site Name* cyclic field.

In addition to the site names defined in the technology file, for IO/Corner cells, the <AUTO> and <NONE> options available in the *Site Name* cyclic field.

Behavior of the <AUTO> option:

- If the layout has an existing site definition, then no default site is created. The site definition from the layout is retained in the final abstract view and is dumped in the LEF file.
- If the layout does not have any site definition, a new default site definition is created and is dumped in the LEF file.

Behavior of the <NONE> option:

- Any existing site definition is removed from the abstract view. So, no site definition is dumped in the LEF file.
- If the layout does not have any site definition, then no new site definition is created. So, no site definition is dumped in the LEF file.

The <NONE> option is not available for CORE cells because CORE cells must have a site definition in LEF file.

To specify sites in the Standalone Abstract Generator:

© 2023 Cadence Design Systems, Inc.

1. Choose *Flow – Abstract*, then click the *Site* tab in the Running step Abstract form.

2. From *Site name*, select a site name defined in the technology library from the available list to be applied to all cells in the current bin.
3. In *Define new site name*, specify the name of a new site to be used by all cells in the current bin.

© 2023 Cadence Design Systems, Inc.

4. Use *Calculate SITE pattern for Gate-array Cells* to derive site patterns from the layout of gate-array standard cells. This option works exclusive of the single site creation fields on the *Site* tab.

5. Specify the sub-cellview name in the *Sub-Cellview* column.
6. In the *site* column, the drop-down list displays the already existing sites. Select one from the list to map the selected sub-cellview.
7. Click *Add* to create the mappings. The *Add* button becomes active when you select the *Calculate SITE pattern for Gate Array Cells* option.

If a gate-array sub-cellview is itself associated with a site pattern, Abstract Generator preserves the pattern and exports it to LEF.

Related Topics

Running step Abstract Form

© 2023 Cadence Design Systems, Inc.

Creating an Overlap Boundary

The overlap boundary is a detailed boundary used by placement tools to test whether cells overlap. Unlike the PR boundary, which is always rectangular, the overlap boundary can be a rectilinear polygon. This helps the placement tools achieve better placement density for certain types of standard cells and rectilinear blocks.

To create an overlap boundary:

1. Choose *Flow – Abstract*, and click the *Overlap* tab in the Running step Abstract for the selected cell(s) form.

   The options on this tab control whether Abstract Generator is to update the overlap boundary geometry.

2. Use the *Create overlap boundary* option to specify whether Abstract Generator should create an overlap boundary.
3. In the *Using geometry on layers* field, specify the layers to be used in the overlap boundary calculation. The boundary is drawn so that it encloses all the geometry found on these layers.
4. In *Size factor to apply*, specify (in microns) the value by which an overlap layer is to be increased.

© 2023 Cadence Design Systems, Inc.

5. In *Smooth factor to apply*, specify the value that controls the shape of the overlap layer that is produced.
6. Click *Run*.

An overlap boundary is created as per the specifications

Creating an Overlap Layer in OpenAccess

As there is no concept of an overlap layer in OpenAccess, Abstract Generator creates the overlap shape on the PR boundary layer. If you export LEF for the cellview, the overlap shape is correctly written to the `OVERLAP` section.

If you create or update the PR boundary by using the options on the *Boundary* Tab and then create an overlap layer by using the options on the *Overlap* tab, the PR boundary is destroyed and replaced with the overlap shape.

Troubleshoot OAToolbox Issues

You can face the following issues while using open access toolbox in Abstract Generator.

ABS-11500

Cell *name*: Row *name* in view *viewName* has a missing property *propertyName* and will not be converted.

ABS-11501

Cell *name*: Row *name* in view *viewName* references a site def *siteDefName* which has not been defined in the technology database.

ABS-11502

Couldn't open master *libName*, *cellName*, *viewName*, referenced in cellview *libName*, *cellName*, *viewName*.

Related Topics

Running step Abstract Form

© 2023 Cadence Design Systems, Inc.

Retain Layout Information in the Abstract View

Abstract Generator allows you to save layout details along with the pin shapes and blockages in the final abstract generated. To retain the layout information, you can use the `AbstractRetainLayout` option of the `absAbstract` SKILL command.

```
absSetBinOption("Block" "AbstractRetainLayout" "true")
```

By default, the `AbstractRetainLayout` option is set to `false`, so the final abstract contains the adjusted pin shapes, blockages, and boundary based on the options specified. All other shapes or data in the layout cell-view is deleted.

To include the layout information in the abstracts, set the `AbstractRetainLayout` option to `true`. The layout information along with the pin shapes and blockages will be written to the abstract view. You can change the name of the abstract view in the *Views Tab* of the *General options* dialog box (*File – General options)* by specifying the name in the *Abstract view name* field.

Alternatively, you can use the `absSetOption` SKILL command line to change the abstract view name.

```
absSetOption("ViewAbstract" "layout.pin")
```

You can use the `AbstractRetainLayout` option to create pins in your layouts so that they can be used in the Virtuoso® XL layout editor environment. If you do not want blockages in

© 2023 Cadence Design Systems, Inc.

these layouts, do not specify any layers in the *Layer Assignment for Blockages* table on the *Blockage* tab of the Abstract step.

Related Topics

General Options Form

© 2023 Cadence Design Systems, Inc.

Troubleshoot Abstract Messages

ABS-14003

Message: Cell *name*: the power ring for net *name* only contains *value* shapes that are within *value* microns of the boundary. There must be at least four shapes within this distance. This is determined using the shape closest to the cell boundary. See Customizing Pin Shapes in Standalone Abstract Generator.

ABS-14006

Message: There are no power and ground rails common to all cells in the current bin. Use the *Power Rail Widths and Offsets* option to specify the characteristics of the power and ground rails for the bin. You might also need to redistribute cells into separate bins to complete this step.

ABS-14007

Message: Cell *name*: contains non-orthogonal geometry which cannot be modeled. There are *value* cells with non-orthogonal geometry.

Description: The design contains one or more cells containing diagonal geometry which Abstract Generator cannot model.

Abstract Generator usually adopts the stair-step coverage method in order to model 45-degree geometry, but this method failed on the named cells. It could be that the intersection of the 45-degree shape and the stair-step is too small to be processed by Abstract Generator.

Troubleshooting: Increase the size of your stair step and run Abstract Generator again. If the problem persists, consider editing your layout data.

- For information about the stair-step model, see Pin and Blockage Fracturing.
- For information about designing library data for use in Cadence flows, see Appendix D, “Library Design Guidelines.”

© 2023 Cadence Design Systems, Inc.

ABS-14008

Message: Cell *name*: terminal *name* shape *name* is different from the calculated value *value*. You can reset shape properties using the Terminal Properties form.

Description: The shape value calculated for the named terminal is different from the value already stored in the cellview. Abstract Generator is unable to reset the property value.

ABS-14013

Message: Cell *name*: The blockage type selected for the layer *name* in the *Layer Assignment for the Blockages* table is *Shrink*. However, no value is specified for either *Shrink Track* or *Shrink Dist*. If you do not specify a value in either of these columns corresponding to the layer, the abstract generator will use the default value of 2 tracks for creating the blockage. This default value is small and will increase the run time of the Abstract step significantly. If you want to avoid this, you must specify either the *Shrink Track* or *Shrink Dist* value or both.

Description: If you choose the type of blockage you want create on a layer as `Shrink` in the *Blockage* column of the *Layer Assignment for Blockages* table on the *Blockage* tab (Abstract step), you need to specify a distance value (in microns) in the *Shrink Dist* field. Alternatively, you can specifiy a value in terms of the number of tracks between the blockages in the *Shrink Track* field. Abstract Generator combines blockages that are within the specified distance from each other or are separated by the specified number of tracks by filling the space between them with more blockage.

If you specify both `Shrink Track` and `Shrink Dist`, the values are added and blockages separated by a distance less than or equal to the total distance are merged. If you do not supply a distance, Abstract Generator uses a default value of `2` tracks and displays this warning message.

ABS-14015

© 2023 Cadence Design Systems, Inc.

Message: Cell *name*: the overlap layer boundary has not been fully enclosed by the prBoundary. Check that the correct layers have been specified in the *Using geometry on layers* field.

Description: There is geometry on the overlap layer which extends beyond the place-and-route boundary.

ABS-14016

Message: Cell *name*: no overlap layer shapes have been generated. Check that the layers have been specified in the *Using geometry on layers* field. See Retain Layout Information in the Abstract View.

ABS-14017

Message: Cell *name*: has more than one overlap shape generated. Check the layers specified in the *Using geometry on layers* field and the abstract view.

Description: The geometry you supplied in the *Using geometry on layers* field in the *Overlap* tab has led to the tool creating more than one shape for the overlap layer.

This error might also appear if the shapes to be considered for creating the overlap boundary exist on the `boundary` purpose, whereas only the layer(s) is specified in the *Using geometry on layers* field.

© 2023 Cadence Design Systems, Inc.

Troubleshooting:

Set the *Smooth factor to apply* to be greater than the distance between the shapes on the overlap layer and rerun Abstract Generator. This forces the tool to create a single shape. See Retain Layout Information in the Abstract View.

To include shapes on the ignored `boundary` purpose, do one of the following:

- Explicitly include the purpose in the geometry specification in the *Using geometry on layers* field.

  For example, if you want the `boundary` shapes on `M1` to be included in the overlap boundary creation, specify the following in the field instead of just `M1`:

  ```
  (M1 boundary) M1
  ```

- Remove `boundary` from the *Ignored Purpose(s)* field on the *General* tab (*File – General options*).

  Since this is a global , updating the Ignored Purpose(s) field in this way will affect all layer definitions. Therefore, choose this alternative only after due consideration.

ABS-14019

Message:

Cell *`name`*: net *`name`* has preset pin geometry which violates minimum area rules. You may want to option re-run the Extract step after ensuring that the *Extract signal nets* and *Extract power nets* options are turned on to create the correct geometry.

Description:

Because full geometry is not being performed, Abstract Generator has created a pin that does not satisfy the minimum area rules set for the layer in the technology file. See *Select the Extract power nets* option to run connectivity extraction for power nets.

ABS-14020

© 2023 Cadence Design Systems, Inc.

Message: Cell *name*: net *name* has boundary pin geometry which violates the minimum area rules. The *Create boundary pins* option has created pins that do not satisfy the minimum area rules in the technology file. You can inactivate the *Create boundary pins* option and rerun the command.

ABS-14021

Message: Cell *name*: net *name* has full shape pin geometry that violates the minimum area rules set in the technology file.

Description: The pin geometry for a given layer must satisfy the minimum area rules defined for that layer in the technology file. If full geometry extraction is performed, and the pin still does not satisfy minimum area rules, it indicates a significant problem in the layout design.

ABS-14022

Message: Cell *name*: geometry has been found abutting or extending past the prBoundary. As a result, the *Create boundary pins* option has been disabled.

Description: Because there is some geometry other than labels, power shapes, or abutments which extends beyond the place-and-route boundary, the *Create boundary pins* option has been disabled automatically. This indicates that there is a problem with the layout design.

ABS-14029

Message: Cell *name*: Trying to create a blockage on layer *layerName*. This layer does not have boundary purpose in the technology data.

Description: This error message is issued when you try to add blockage(s) on a layer that does not have a purpose defined in the technology file.

Troubleshooting: Use the Technology File Manager to define the missing LPPs in the `techLayerPurposePriorities` section of the technology file. Also see ABS-10050.

© 2023 Cadence Design Systems, Inc.

ABS-14030

Message: Cell *name*: An internal error occurred while performing metal density calculation. *Additional information*.

Troubleshooting: This is an uncommon error message. Look for warning messages received prior to recieving this error message and try to resolve the problem. If you get stuck, contact Cadence support with the problem testcase.

ABS-14031

Message: Cell *name*: An internal problem occurred while performing metal density calculation for the *layer name*. *Additional information*.

Troubleshooting: This is an uncommon error message. Look for warning messages received prior to recieving this error message and try to resolve the problem. If you get stuck, contact Cadence support with the problem testcase.

ABS-14032

Message: Cell *name*: Density data will not be calculated for the non-metal layer *layer name*.

Description: This warning message appears if you are using the *Layer Assignment for Metal Density Regions* table on the *Density* tab for density calculation and have specified a non-metal layer in the table. Abstract Generator will ignore the non-metal layer for density calculation.

ABS-14033

Message: Cell *name*: You are calculating metal density for the *layer name* layer. The technology file does not have a minimum density rule for this layer.

© 2023 Cadence Design Systems, Inc.

Description: This warning message appears when you try to calculate density for a metal layer for which no minimum density rule has been defined in the technology file. In the absence of the minimum density rule for the metal layer, the density information generated would not be useful.

To add minimum density rule for a metal layer in the technology file, see *Virtuoso Technology Data ASCII Files Reference*.

ABS-14035

Message: Cell *name*: PR boundary is not rectilinear. Edges at non-orthogonal angles will be approximated to larger rectangular shapes, which might overlap other such rectangular shapes. Stair step approximation is currently not supported.

Description: Metal density calculations are based on the dimensions of rectangular density windows. However, when the edges of a PR boundary are non-orthogonal, Abstract Generator will extrapolate the shape to the closest possible rectangle. The resulting rectangular shape encloses regions outside the PR boundary and can overlap other such approximated rectangles.

Troubleshooting: Check the generated abstract to ensure that it meets the required guidelines.

ABS-14036

Message: Cell *name*: does not have a valid PR boundary. The bounding box of the cell view will be used to generate density window(s).

Description: Reconsider the values selected for the *Create boundary* and *Create overlap boundary* options on the Boundary tab (Pins step) and Overlap tab (Abstract step), respectively, to obtain a valid PR boundary for the cell.

ABS-14037

Message: Cell *name*: Site pattern for the gate-array cell *name* cannot be calculated because it contains an instance of the Pcell *name*. Site pattern calculation for Pcells is not supported.

© 2023 Cadence Design Systems, Inc.

ABS-14038

Message: Cell *name*: Site pattern for the gate-array cell *name* cannot be calculated. The gate-array cell contains an instance of the sub-cellview *name*, which itself is associated with a site array.

Description: The gate-array cell contains an instance of the sub-cellview name, which itself is associated with a site array.

ABS-14039

Message: Cell *name*: The sub-cellview *name* is mapped to the site *name* in the *Sub-Cellview to Site Mapping* table. However, this sub-cellview is already option attached to the site *name*. The mapping entry specified in the table will be ignored.

Description: The sub-cellview's association to a site is different from what is specified in the *Sub-Cellview to Site Mapping* table on the *Site* tab. Therefore, the mapping entry specified in the table will be ignored.

ABS-14040

Message: Cell *name*: Site pattern for the gate-array cell *name* cannot be calculated. The gate-array cell contains an instance of the sub-cellview *name*, which is not mapped to a site in the *Sub-Cellview to Site Mapping* table. Ensure that the sub-cellview to site mapping is defined in the *Sub-Cellview to Site Mapping* table on the *Site* tab.

ABS-14041

Message: Cell *name*: Site pattern for the gate-array cell *name* cannot be calculated. The site *name* mapped to the gate-array cell *name* in the *Sub-Cellview to Site Mapping* table does not exist. Ensure that the site to which the sub-cellview is mapped is either predefined or already exists.

© 2023 Cadence Design Systems, Inc.

Related Topics

Customizing Pin Shapes in Standalone Abstract Generator

Pin and Blockage Fracturing

© 2023 Cadence Design Systems, Inc.

Creation of Fill Blockages on Specific Layers

All detailed or cover blockages created in Abstract Generator are exported under the `OBS` section and all fill blockages under the `DENSITY` section of an LEF file. You can create fill blockages automatically on specific layers based on the detailed blockages generated in Abstract Generator.

By default, fill blockages are generated when the *Calculate metal density* option is turned on under the *Density* tab in the Abstract step. The following options can be set in the `abstract.replay` file or the `.abstract.options` file for creating fill blockages.

```
absSetBinOption("Core" "AbstractDensity" "true")
absSetBinOption("Core" "AbstractDensityLayers" "Metal1 Metal2 Metal3")
```

You can use the `AbstractPostHook` function to automatically create fill blockages on specific layers. This function is triggered after the creation of the abstract view, and it uses the point lists of the detailed or cover blockages that are generated. The fill blockages are then created on layers as per the `LayersList` variable defined in the SKILL code. You can also change the `blockageType` option to create other blockages such as feedthru, slot, and screen in the abstract view.

© 2023 Cadence Design Systems, Inc.

Example: Type the following SKILL `AbstractPostHook` code at the `abstract` prompt.

```
procedure(AbstractPostHook(binName libName cellName abstractViewName)
let((cv LayersList blockageType RoutingBlockages)
cv = dbOpenCellViewByType(libName cellName abstractViewName "maskLayout" "a")

LayersList=list("Metal4" "DIOdummy")   ;; change the list of layers on which extra
blockages are to be created
blockageType="fill"                    ;; Change the blockage type to any fill,
screen, feedthru etc.

RoutingBlockages=setof(x cv~>blockages x~>type=="routing")

foreach(blockage RoutingBlockages
    foreach(layer LayersList
          printf("Creating new Fill Blockage from layer = %s to layer = %s at %L\n"
            blockage~>layer layer blockage~>points)
            dbCreateLayerBlockage(cv layer blockageType blockage~>points)
    );foreach
);foreach
dbSave(cv)
dbClose(cv)
) ;let
) ;procedure
```

Related Topics

Density Calculation

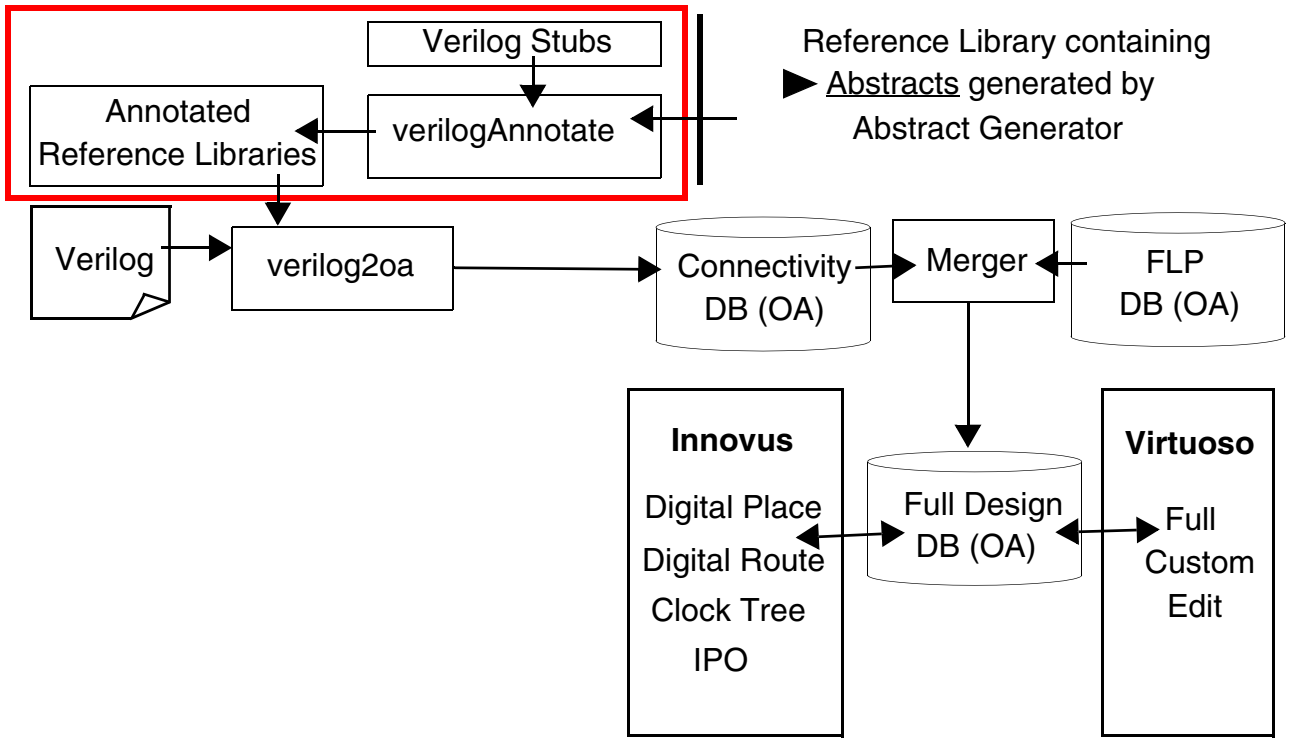
© 2023 Cadence Design Systems, Inc.

Bus Annotation in Abstract Generator

Bus Annotation for Abstract Generator improves the interoperability between Virtuoso and Innovus. You can directly import the library elements created inside Abstract Generator to Innovus.

In OpenAccess, cellview data of a library element created by applications such as Abstract Generator (AG) usually consists of explicit scalar bits (A<0>, A<1>) corresponding to a vector bus terminal (A<0:1>). This is because pins and corresponding pinFigs have to be created on each bit with specified layer and shape for physical connectivity.

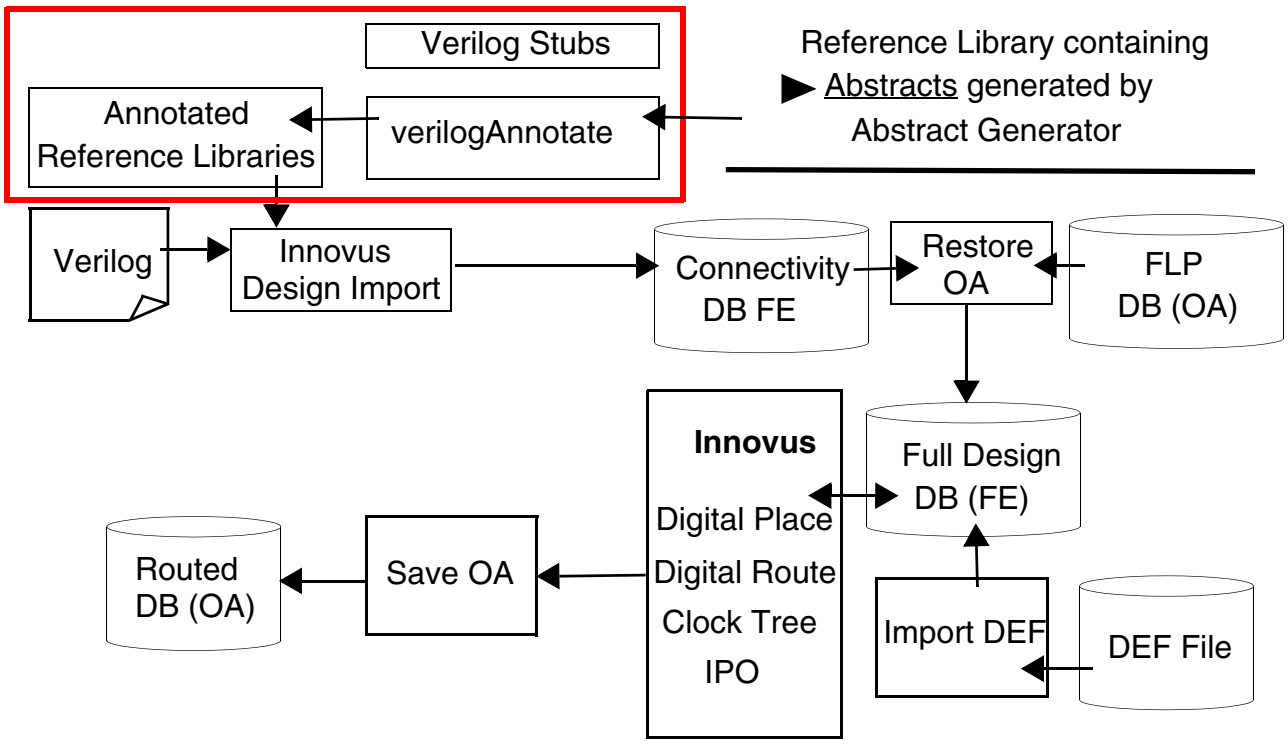
When OA cellview of library element is used by applications like Innovus and verilog2oa, as shown in the following two figures, the order of bits on a bus, which can be either ascending (0:1) or descending (1:0), is important. This cannot be inferred from scalar bits present in the OA data of the library element.



For this flow to work, run Verilog Annotate on the library elements before data is used in Innovus. The verilogAnnotate annotates the vector bus terminal information on the OA cellview of the library element. Running verilogAnnotate requires a corresponding interface definition in verilog format of the module corresponding to the library element. For each library element, a verilog stub file is created as described in the following figure. The verilog stub file is created from corresponding schematics and symbol definitions by using the Create –

© 2023 Cadence Design Systems, Inc.

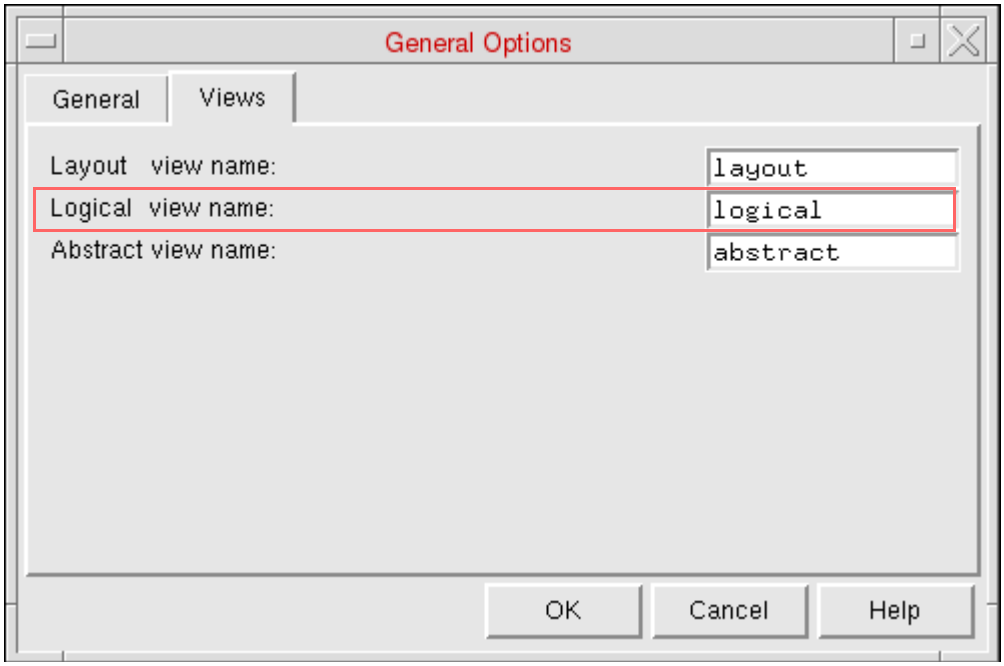
Cellview – From Cellview commands followed by Verilog Annotate. However, creating verilog stubs and running `verilogAnnotate` is a time consuming process.



In Virtuoso XL, while creating an abstract, an input logical view option can be used to specify the symbol, schematic, verilog, or .liberty (.lib) from which bus definition is extracted. The vector terminal information is directly added to the OA cellview of the library element by the

© 2023 Cadence Design Systems, Inc.

Abstract Generator engine and there is no need to generate the verilog stub file and run verilogAnnotate.



Support for EXCEPTPGNET LEF Attribute

The EXCEPTPGNET lef attribute is mapped to the OA attribute allowPGNet of oaLayerBlockage object. To support the EXCEPTPGNET LEF attribute in LEF5.8, the following bin specific option has been added in Abstract Generator.

```
absSetBinOption( "Core" "AbstractBlockageEXCEPTPGNET" "Metal1 Metal2")
```

When the `AbstractBlockageEXCEPTPGNET` option is set as specified in the above example, then all the blockages generated by Abstract on Metal1 and Metal2 will have EXCEPTPGNET lef attribute in the lef file.

This option is set for the blocks that want to block routing on a given layer except for power and ground routing. This is usually true for analog or memory blocks that are sensitive to noise, and do not want to allow signal routing on the layers immediately above the block as the intention is to allow power routing on the layers.

Related Topics

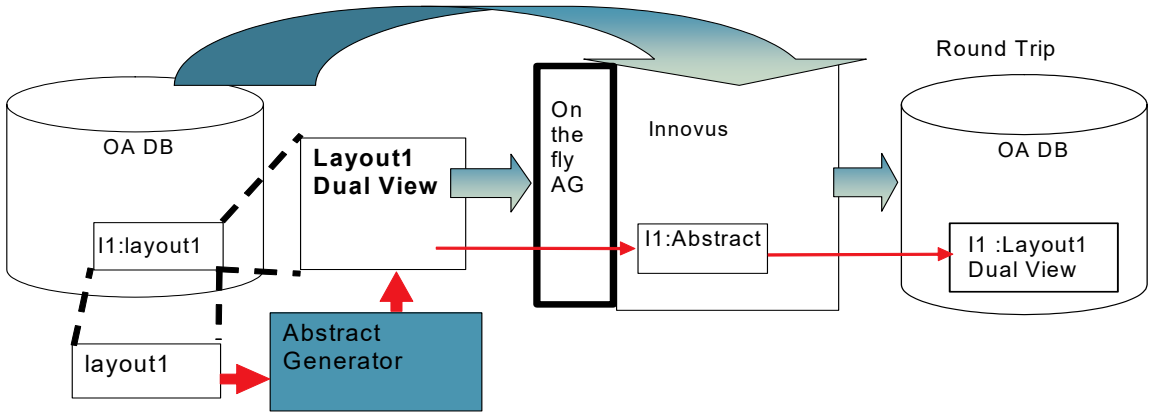
Abstract Generator Options for Layout Dual View

Create – Cellview – From Cellview

© 2023 Cadence Design Systems, Inc.

Layout Dual View Support

A layout dual view contains both abstract and layout information. Innovus requires Abstract Generator to generate a partially annotated layout dual view. As a result, Abstract Generator annotates the required abstract information in the layout view and will not generate a separate abstract cellview. The following figure shows how a layout dual view is used in the Innovus flow.



Each of the three run steps of Abstract Generator—*Pins*, *Extract*, and *Abstract*—can generate the layout dual view.You need to run only the *Pins* step because the antenna, extraction, and blockage information (generated by Extract and Abstract steps) is not required in the layout dual view. When you run the *Pins* step, Abstract Generator saves a valid view of pins along with the layout dual view so that the *Pins* view can be used by the *Extract* and *Abstract* steps, if required.

The advantages of layout dual view are as follows:

- Innovus no longer needs to maintain the abstract and layout views separately. This helps overcome the problem of maintaining two similar views for layouts that are under development.
- While routing, the router can directly use the layout dual view without the need to master and re-master the layout. This helps increase the productivity.

Related Topics

Abstract Generator Options for Layout Dual View

Abstract Data Annotated in Layout Dual View

© 2023 Cadence Design Systems, Inc.

Abstract Data Annotated in Layout Dual View

The abstract data that is annotated in the layout dual view is as follows.

- **prBoundary**

  The prBoundary of the layout dual view is updated if it already exists in the layout view; otherwise, it is created based on the *Create boundary* options on the *Boundary* tab of Abstract Generator.

- **cellType**

  If the cellType is already specified for the layout, Abstract Generator does not modify the cellType in the layout dual view. If the cellType attribute is not specified, Abstract Generator automatically sets it depending on the bin in which the cell is processed.

- **symmetry**

  Abstract Generator uses the following sequence to determine the symmetry of the current cellview:

  - The *Symmetry Type* attribute in layout cellview is considered.
  - If the *Symmetry Type* attribute is not set in layout cellview, then the property with the name *Symmetry* on layout cellview is considered.
  - You can overwrite the above values by selecting a value in the *symmetry* field on the Cell Properties form.

- **timestamp**

  The timestamp of the annotated abstract data is updated in the layout dual view. This is done to verify whether or not the abstract data in the layout dual view is up to date.

- **Cover blockage**

  A cover blockage is created for a PR boundary in the layout dual view with the value provided by the *AbstractTopLayerCoverBlockage* option in Abstract Generator Options for Layout Dual View. Alternatively, you can also create cover blockages using the *Specify the top metal layer for cover blockage* option on the *Map* tab of the *Pins* step.

- **Pin Annotation**

  Abstract generator enables you to create pins from the text labels by using the *Map text label to pins* option in the Pins step. You can also create pins from the text labels by setting the `PinsTextPinMap` option, as follows:

  ```
  absSetBinOption("Core" "PinsTextPinMap""(M1 (M1 label))")
  ```


© 2023 Cadence Design Systems, Inc.

The new pins created by Abstract Generator based on the pin text label map are annotated in the layout dual view.

If the layout has existing pins and pin text label map is set, only the new pins added by Abstract Generator are annotated in the layout dual view. If the layout has existing pins and pin text label map is not set, no new pins are created and the existing pins are preserved in the layout dual view.

Abstract Generator Options for Layout Dual View

For generating the layout dual view, the following two new options have been added to Abstract Generator.

- **AnnotateLayoutDualView**

  To enable Abstract Generator to generate the layout dual view, set the `AnnotateLayoutDualView` option to `TRUE`. If this option is set to `TRUE`, Abstract Generator annotates the abstract data in the layout view to generate the layout dual view. By default, this option is set to `FALSE`.

  ```
  absSetOption("AnnotateLayoutDualView" "true")
  ```

- **AbstractTopLayerCoverBlockage**

  This option specifies the top metal layer for cover blockage in the layout dual view. This new option has been added because the existing cover blockage options have different use model, which may lead to confusion.

  ```
  absSetBinOption("Core" AbstractTopLayerCoverBlockage" "Metal5")
  ```

Related Topics

Pins Step in Standalone Abstract Generation

Cell Properties Form

Edit Cellview Properties Form

Troubleshoot Layout Messages

© 2023 Cadence Design Systems, Inc.

Troubleshoot Layout Messages

ABS-11804

You have selected one of the extraction options (signal, power, or antenna extraction) and there are shapes on purpose `boundary` in the layout view for at least one of the layers entered in an extract geometry specification (GSpec) table. For some designs, you do not need to take any action. However, if your design contains large shapes on purpose `boundary`, this can cause extraction problems (such as shorts in a design) which are only discovered after a protracted extraction process.

You have chosen to perform either signal, power, or antenna extraction, but there are shapes on purpose boundary in your layout view. Large boundary shapes can cause extraction problems, such as shorts, but can be remedied by excluding the purpose boundary by using *General options* dialog box or by entering the following (example) geometry specification: `metal1 -> (metal1 drawing) or (metal1 pin) or (metal1 net)` in the appropriate GSpec location in the Extract step.

Troubleshooting: Remove purpose `boundary` from the geometry specification. For example, instead of specifying `metal1`, type the following instead.

```
(metal1 drawing) or (metal1 pin) or (metal1 net)
```

Even if you do not perform extraction, the pins created in the Abstract step could still be affected if you have blockage shapes on the layout.

Suppress Messages Issued by Abstract Generator

Use the `suppressMessage` option to suppress INFO and WARNING messages issued by Abstract Generator.

```
absSetOption(suppressMessage <id1> <id2> <id3> …)
```

This option suppresses the messages with the mentioned message IDs. ERROR messages are not suppressed. If any ERROR message ID is mentioned in `suppressMessage`, then a warning is issued stating that this is an ERROR message and will not be suppressed.

For every suppressed message, an INFO message is displayed that this message is suppressed and will not be shown. This INFO message is displayed only once. At the end of the Abstract step, a table shows the messages that have been suppressed in the current run, and this information is displayed for every cell.

Related Topics

Abstract Data Annotated in Layout Dual View

© 2023 Cadence Design Systems, Inc.

Layout Dual View Support

© 2023 Cadence Design Systems, Inc.

Support for Copying Mixed Signal Routing Constraints from Layout to Abstract View

In a layout design, you can create mixed signal routing constraints on a block using the Virtuoso Constraint Manager. When an abstract is created for a block from a layout design, the abstract view does not contain the constraint information of the block.

Abstract Generator provides support for copying mixed signal routing constraints from the layout view to the abstract view. In this release, the following mixed signal routing constraints are supported.

- `ciDiffPairType`
- `ciShieldingType`
- `ciNetBusType`
- `ciSymmetryType`
- `ciProcessOverrideType`
- `ciMatchedLengthType`

You can copy the option signal routing constraints from the layout view to the abstract view, either while a new abstract is being generated or to an existing abstract, as explained below:

- Copying constraints to new abstracts during abstract generation

  To copy mixed signal routing constraints to the new abstracts that are created, set the `CopyMSRoutingConstraints` option to `TRUE`. See `CopyMSRoutingConstraint`.

  In addition, for a constraint to be copied to the abstract view, the following three conditions must be satisfied:

  - The routing constraints must be present in the top-level layout view
  - The routing constraint must be present in a net that is part of the abstract
  - The routing constraint must have the Hierarchical Scope attribute set to *above*

  Hierarchical scope is an attribute of a routing constraint that can have the following values: local, below, and above. The value ‘local’ is always set for a routing constraint.

  By default, the value of the Hierarchical scope attribute is set to local. Therefore, ensure that you set it to above before you create the abstract from the layout.

- Copying constraints to existing abstracts without regenerating the abstracts

© 2023 Cadence Design Systems, Inc.

You can add mixed signal routing constraints to existing abstracts in VLS XL and VLS GXL. To add constraints, open the existing abstract view in the Virtuoso Layout window and use one of the following methods:

- Load the constraints from the specified connectivity source view by choosing *Connectivity – Update – Layout Constraints.*
- Copy the constraints from a layout view by using the *Load Physical View* command.
- Add constraints to an existing abstract by using the Routing constraints available in the Constraint Creation list in the Constraint Manager.

Related Topics

Updating Layout Constraints

Load Physical View

Constraint Manager Assistant

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

6

Integrated Abstract Generator

The integrated Abstract Generator lets you generate abstracts while you are working on layout designs in the Virtuoso Studio design environment. It is a subset of the standalone Abstract Generator tool, with the minimum and mandatory user interface options. It is presented as a form in which all the abstract steps are available, segregated through progressive disclosure buttons, with the most frequently used options as part of main Abstract Generator flow steps.

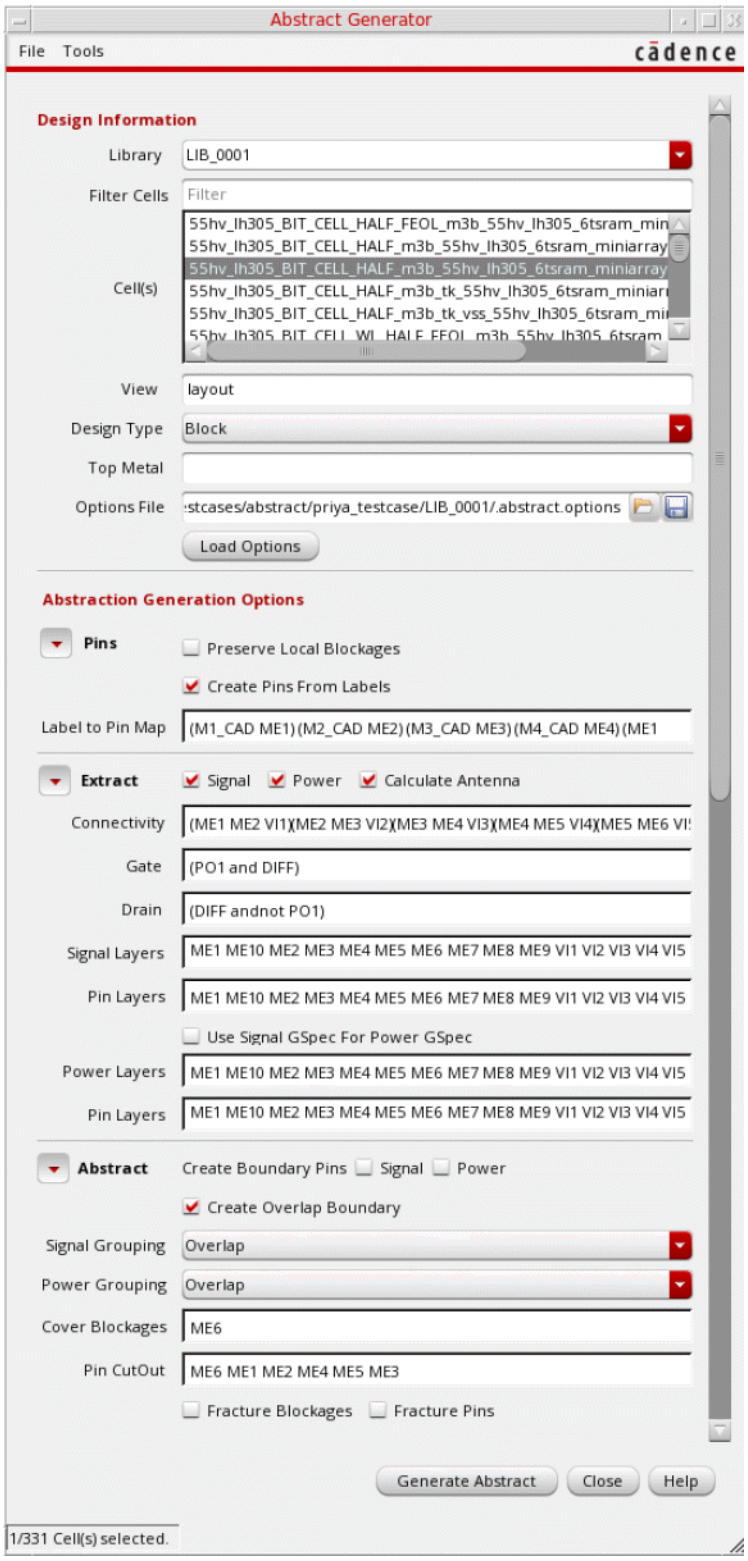
The integrated Abstract Generator in the Virtuoso Studio design environment is a simple interface to generate abstracts for one or more cellviews.

To launch the integrated Abstract Generator interface, do the following:

➥ In the CIW, choose *Tools - Abstract Generator*.

© 2023 Cadence Design Systems, Inc.

The Abstract Generator form is displayed.



© 2023 Cadence Design Systems, Inc.

You can load values in the Abstract Generator form, run the Abstract Generator steps and view the results during the abstract generation process.

Related Topics

iagGenAbstract

Generating Abstracts

Viewing Results in the Abstract Generator Form

Customizing Abstract Generator Flow Using SKILL Files

Integrated Abstract Generator Form

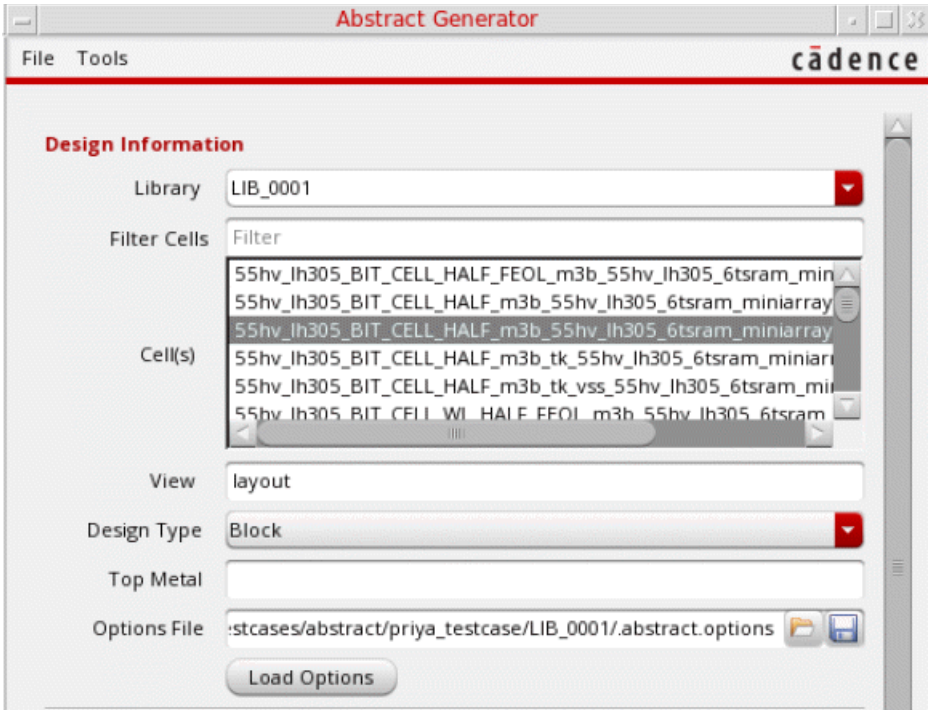
© 2023 Cadence Design Systems, Inc.

Specifying Design Information in the Abstract Generator

You can load values in the Abstract Generator form, run the Abstract Generator steps and view the results during the abstract generation process. To initialize the fields in the Abstract Generator form:

1. In the CIW, choose *Tools - Abstract Generator*.

   The Abstract Generator form is displayed.



2. Select a library.

   The library list is read in from the `cds.lib` file located in the run directory, and libraries are listed in an alphabetical order. When you select a library, the *Cell(s)* list is automatically populated.

3. In the *Filter* Cells field, you can filter out cells based on their names. For example, type `top` to filter out cells that contain the string in their name.

4. From the *Cell(s)* list, select the required cells. Use one of the following selection methods:

   - Select one or multiple cells by clicking their names.

© 2023 Cadence Design Systems, Inc.

- Select multiple consecutive cells by clicking a cell and dragging the cursor across all the cells to be selected without releasing the mouse button. Release the mouse button after all the required cells have been selected.
- Select all cells by using the `Ctrl+A` bindkey.

The number of cells selected is displayed in the status bar.

To deselect cells:

- Click the individual cells to be deselected.
- To deselect all the cells at one go, use the `Ctrl+D` bindkey.

5. In the *View* field, specify the view name in which you want to create abstracts.
6. From the *Design Type* drop-down list, select a design type. Design type indicates the type of cells to be processed in this abstract run: *Block* (default), *Core, IO*, or *Corner*.
7. (Optional) Specify a *Top Metal* layer. If you specify a top metal layer, the layers listed in the *Extract* and *Abstract* steps are automatically restricted to the specified top metal layer. This means that abstract shapes are generated only for objects on the specified layer and its lower layers, and not for any others. For example, if *Top Metal* is set to `M3`, then the other layer options are reset to include only layers `M3` and its lower layers. Abstract shapes are generated only for objects on these layers.
8. In the *Options File* field, specify the path from where Abstract Generator loads the options.
9. Click *Load Options*.

Abstract Generator loads the options file in the following order:

- The options file (`.abstract.options`) from the *<current working directory>*`/.abstract/`*<library name>* directory is loaded. If not available in the above location, Abstract Generator reads the file first from the attached technology library directory and then from the design library directory.
- The file specified in the *Options File* field in the Abstract Generator form is loaded. If the same options are available in the next file loaded, the options are overwritten accordingly.

Related Topics

Specifying Design Information in the Abstract Generator

Generating Abstracts

© 2023 Cadence Design Systems, Inc.

Viewing Results in the Abstract Generator Form

© 2023 Cadence Design Systems, Inc.

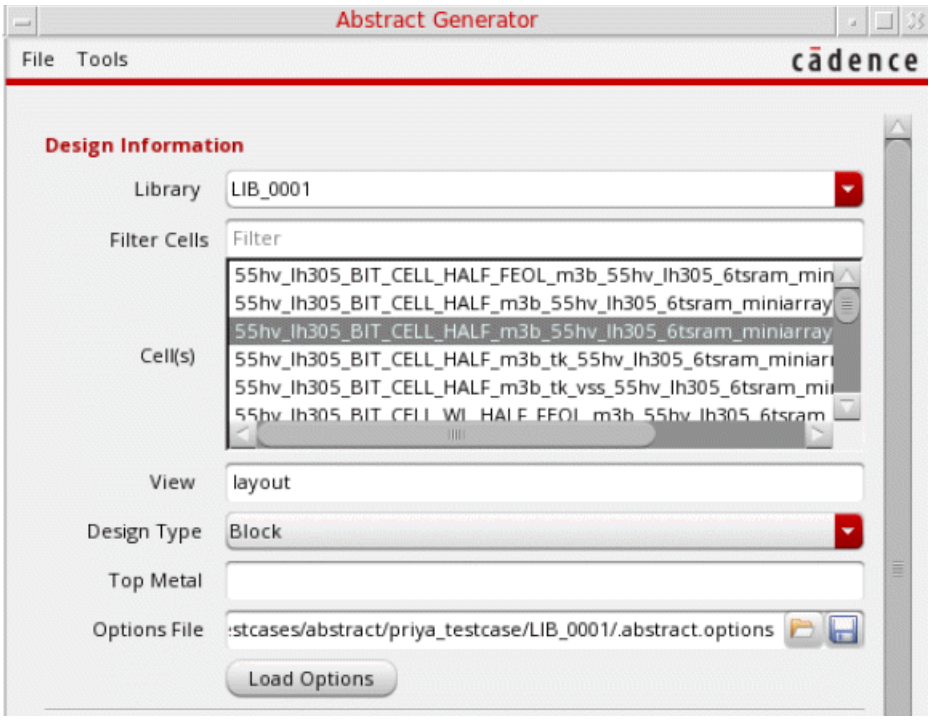
Generating Abstracts

You can load values in the Abstract Generator form, run the Abstract Generator steps and view the results during the abstract generation process. When all the options are loaded, the Abstract Generator form features separate sections for each of the abstract generation steps.

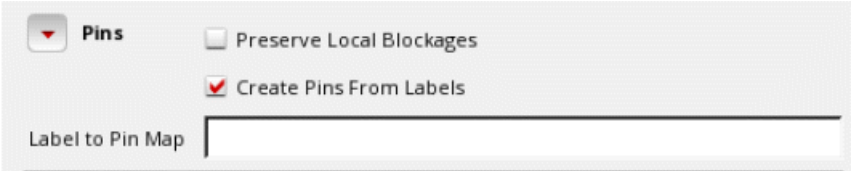
To generate abstracts:

1. In the CIW, choose *Tools - Abstract Generator*.

   The Abstract Generator form is displayed.



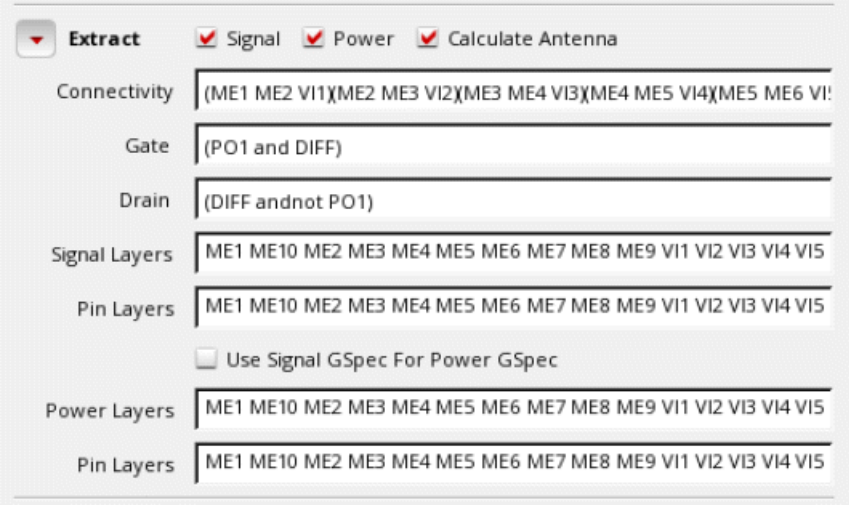
2. Initialize the fields in the *Design Information* section and load the options file.
3. In the *Pins* section, you can use the progressive disclosure arrow to reveal or hide the options.



   The integrated Abstract Generator preserves local blockages and creates pins from labels during the *Pins* step.

© 2023 Cadence Design Systems, Inc.

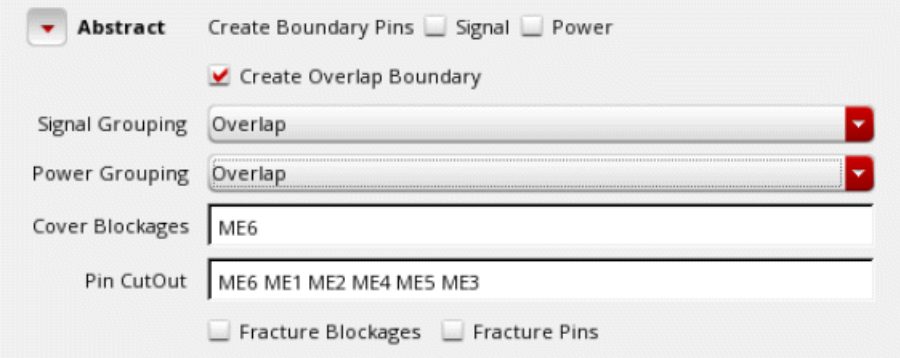
4. In the *Extract* section, you can extract new pins by tracing the physical connections starting from pins created in the *Pins* step. Antenna numbers are also calculated based on the options selected. You can use the progressive disclosure arrow to reveal or hide the options.



- ❑ Choose whether you want to extract connectivity information for *Signal*, *Power*, or both types of nets.
- ❑ Select *Connectivity* to specify the connectivity between layers. You can specify the layer connectivity without having to add via definitions in the technology file.
- ❑ Select *Calculate Antenna* to create process antenna information for custom blocks and standard cells.
- ❑ Select *Gate* and *Drain* text boxes to assign layers to antenna regions to identify the gate and drain regions and to enable antenna calculation.
- ❑ Select *Signal Layers* to list all signal layers to be extracted. Select *Use Signal GSpec for Power GSpec* to extract the geometry specifications for these layers.
- ❑ Select *Pin Layers* to list all the layers on which pins need to be created for signal nets.
- ❑ Select *Power Layers* to list all power layers to be extracted. Select *Use Signal GSpec for Power GSpec* to extract the geometry specifications for these layers.
- ❑ Select *Pin Layers* to list all the layers on which pins need to be created for power nets.

© 2023 Cadence Design Systems, Inc.

5. Abstract is the third and final step of the abstract generation process. You can use the progressive disclosure arrow to reveal or hide the options.



   - Use *Create Boundary Pins* to create square pins at the boundary from the extracted pin geometries. You can create boundary pins for *Signal* nets, *Power* nets, or both.
   - Select *Create Overlap Boundary* to specify whether a rectilinear PR boundary (overlap) should be generated during abstraction.
   - Select *Signal Grouping* and *Power Grouping* to control how the pin geometries are grouped for corresponding signal and P/G nets.
   - In the *Cover Blockages* field, specify the layers to be considered while creating cover blockages.
   - In the *Pin CutOut* field, specify the layers on which pin cutouts need to be created. Pin cutouts are applicable for all types of blockages: cover, detailed, and shrink.
   - Select *Fracture Blockages* and *Fracture Pins* options to generate only rectangular geometries for pins and blockages in the final abstract view.

6. When you are satisfied with the options settings, use the above four flow steps to generate abstracts for the selected cells. You can run the steps either one at a time or all at once for any or all of the cells.
7. After updating values in the *Design Information* and *Extract* sections, click *Generate Abstract* to run Abstract Generator.

The Abstract Log window is displayed.

Related Topics

Specifying Design Information in the Abstract Generator

© 2023 Cadence Design Systems, Inc.

Generating Abstracts

Viewing Results in the Abstract Generator Form

© 2023 Cadence Design Systems, Inc.

Viewing Results in the Abstract Generator Form

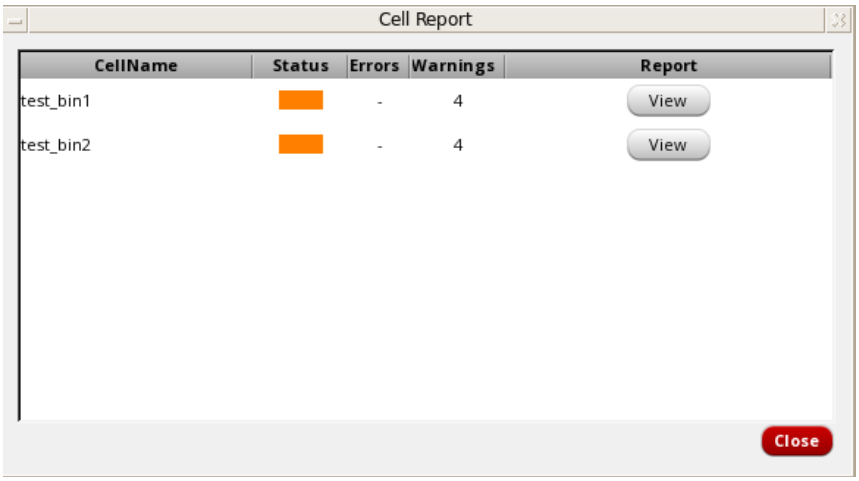
You can load values in the Abstract Generator form, run the Abstract Generator steps and view the results during the abstract generation process. You can use the log window to check whether the results are satisfactory. You can also use the Cell Report window to check the status of the cells that were included in the abstract generation process.

Viewing the Cell Report

To view the cell report:

1. In the Abstract Generator form, choose *Tools – Show Cell Report*.

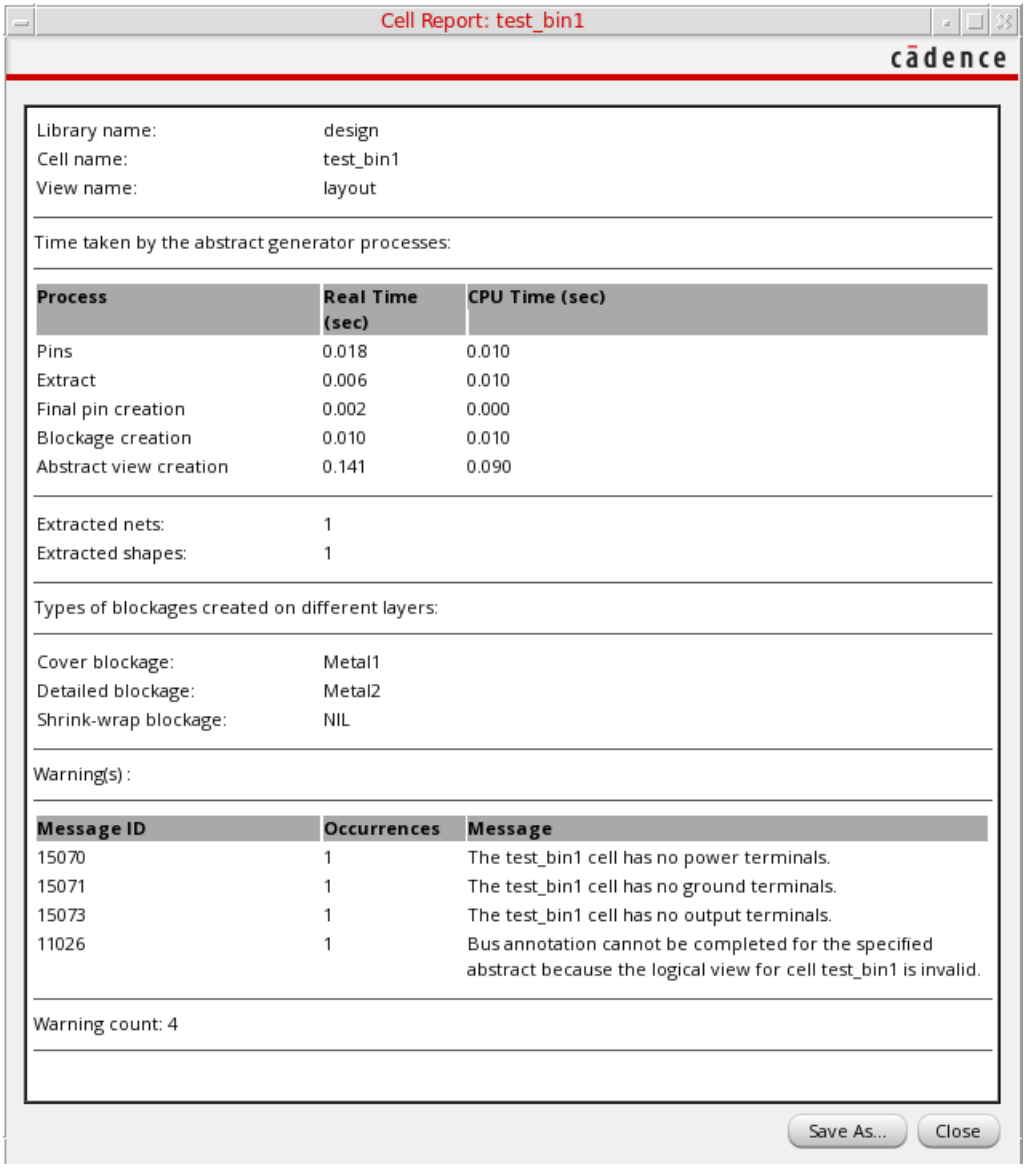
   The Cell Report window is displayed.



2. Click the *View* button to view detailed information about the abstract generation process, including the time taken for abstract generation, types of blockages created on different

© 2023 Cadence Design Systems, Inc.

layers, and the complete list of errors and warnings that were reported.



Viewing Log Information

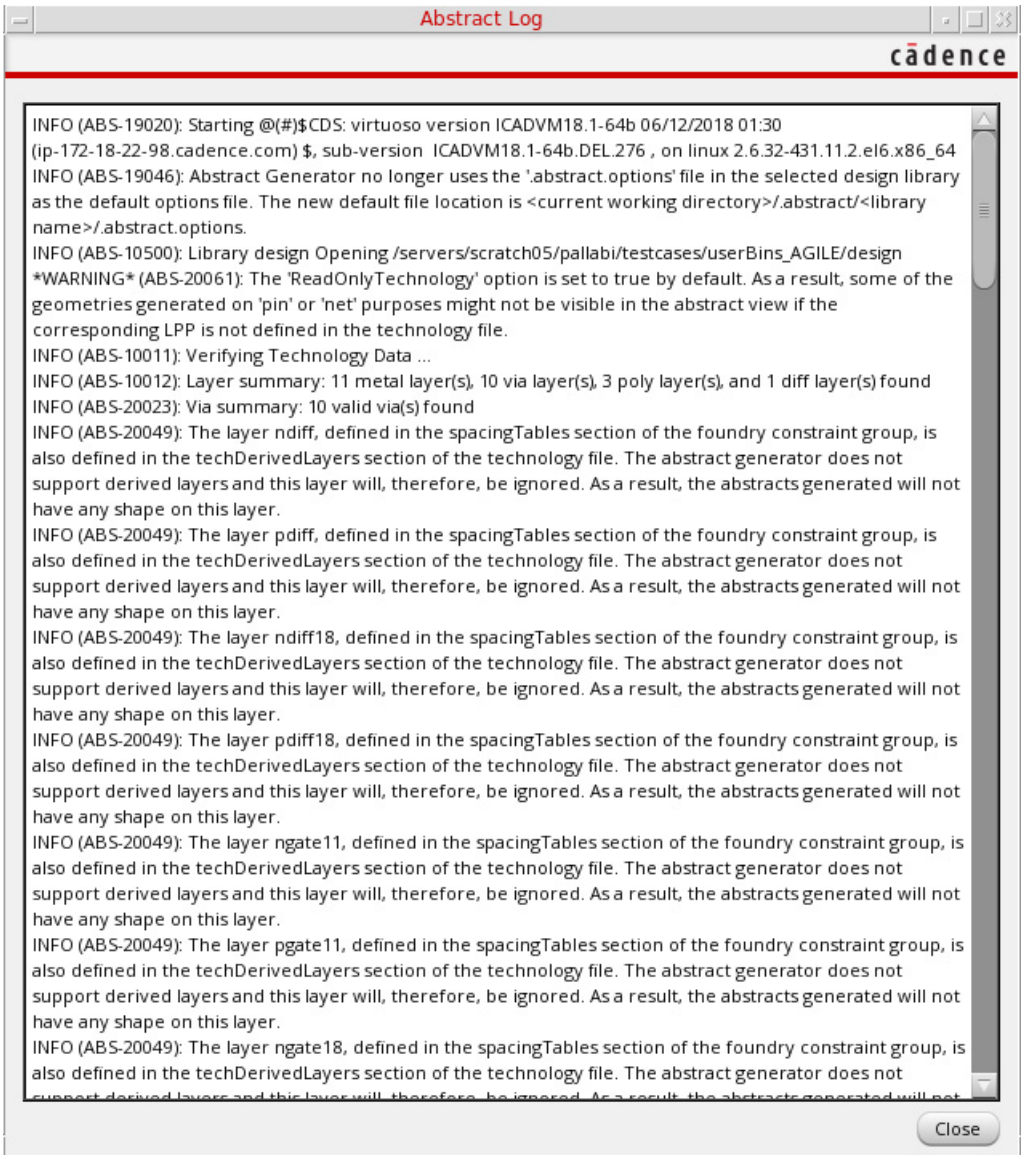
The Abstract Log window displays runtime information, including all the information, warning, and error messages issued by Abstract Generator during the current session.

To view log information,

➡ In the Abstract Generator form, choose *Tools – Show Abstract Log*.

© 2023 Cadence Design Systems, Inc.

The Abstract Log window is displayed.



The log information is also written to the `CDS.log` file.

Viewing the List of Cells

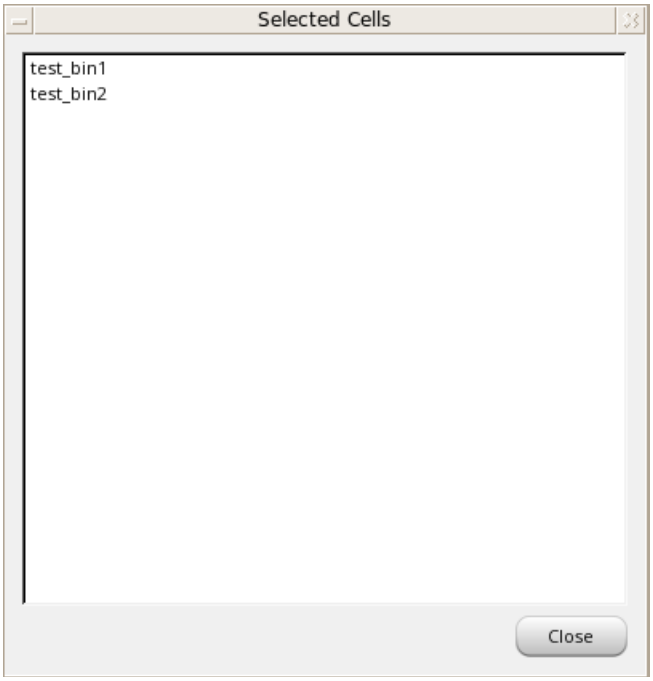
The Selected Cells window displays a list of all the cells that are selected in the *Cell(s)* field in the *Design Information* section of the Abstract Generator form.

To view the selected cells,

© 2023 Cadence Design Systems, Inc.

➡ In the Abstract Generator form, choose *Tools – Show Selected Cells*.

The Selected Cells window is displayed.



Related Topics

Cell Properties Form

Specifying Design Information in the Abstract Generator

Generating Abstracts

© 2023 Cadence Design Systems, Inc.

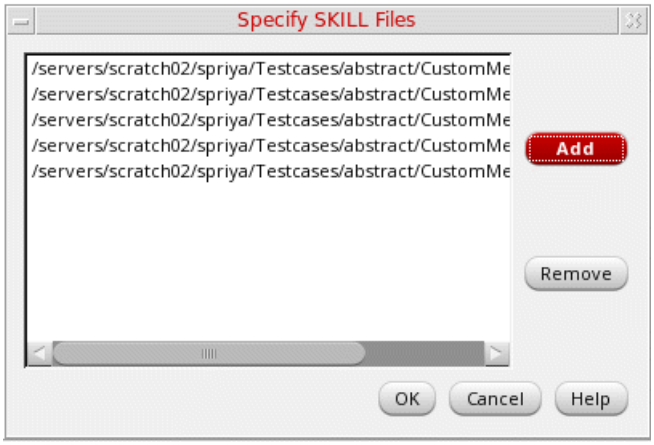
Customizing Abstract Generator Flow Using SKILL Files

You can use custom SKILL code to customize the Abstract Generator flow for specific tasks that cannot be performed using the options in the Abstract Generator form. The custom code, written in a SKILL file, is also called a SKILL hook file.

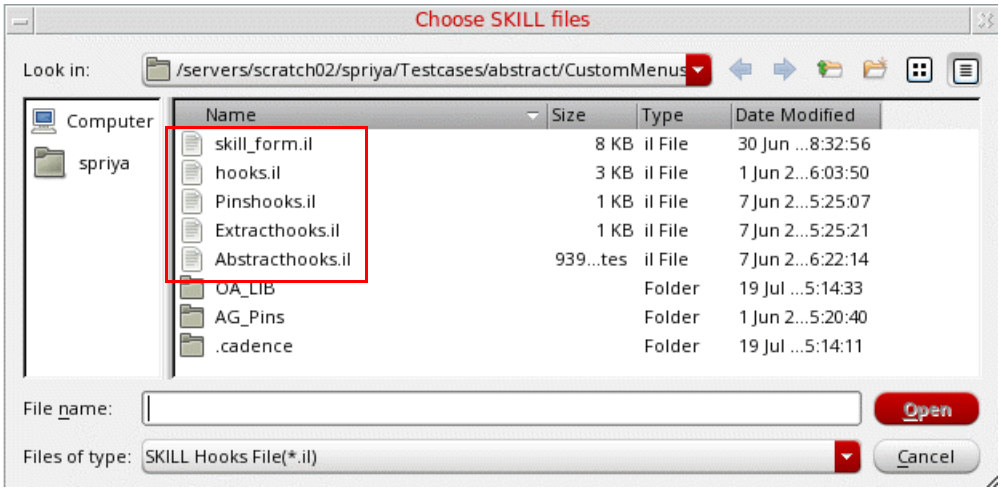
After creating the required SKILL hook files, select the SKILL hooks to run during the abstract generation flow as follows:

1. In the Abstract Generator form, choose *Tools – Specify SKILL Files*.

   The Specify SKILL Files form is displayed.



2. Click *Add* to display the Choose SKILL files window.



3. Select the required SKILL hook files and click *Open*.

© 2023 Cadence Design Systems, Inc.

The selected files are listed in the Specify SKILL Files form.

Note: To remove a file from the list, select the file and click *Remove*.

4. Click *OK*.

The selected SKILL hook files are run as part of a customized abstract generation flow when you click *Generate Abstracts* in the Abstract Generator form. You can remove one or more SKILL files from the subsequent runs in case they are not required.

Related Topics

Customization of the Abstract Generation Flow

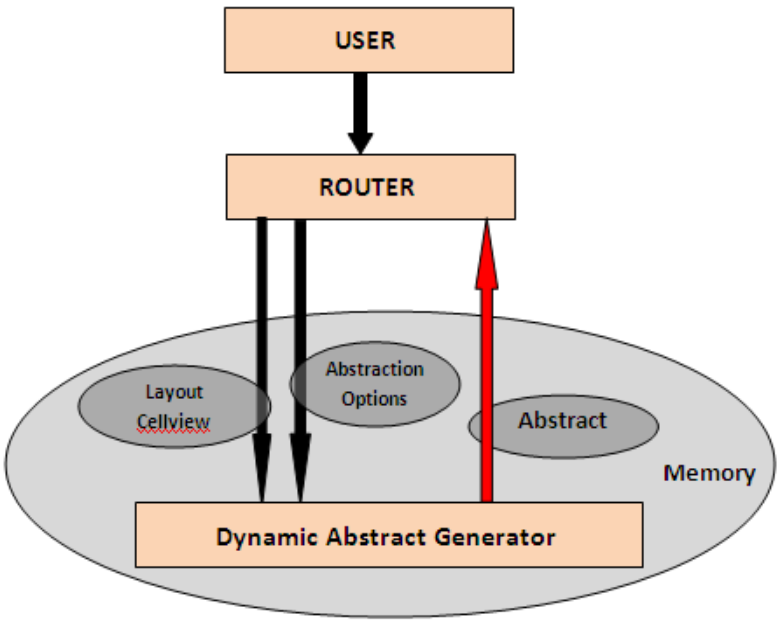
© 2023 Cadence Design Systems, Inc.

7

Dynamic Abstract Generation

Dynamic Abstract Generation automatically creates abstracts for the Pcells based on a set of abstraction rules. The abstract data is stored in memory and is annotated to the Pcell sub-masters corresponding to the Pcells in the design. Because an annotated Pcell submaster contains abstract data annotated over the layout geometries, it is called the dual view. The dual view can be used directly by the router without the need to master and remaster the layout.

Dynamic Abstract Generation is used by Virtuoso Space-Based Router and Create Wire.



Dynamic Abstract Generation has been developed specially for device-level routing— the blockage information that the abstracts contain helps avoid routing over devices. The following additional capabilities of Dynamic Abstract Generation also helps in routing:

- Creates an accurate place and route boundary (prBoundary)
- Creates detailed blockages by default; also provides the option to create cover blockages

© 2023 Cadence Design Systems, Inc.

- Creates and resizes pins and preserves existing pins; pins can also be created from text labels
- Identifies strongly connected pins, weakly connected pins, and must-join pins
- Extracts nets by using the layer-geometry specification

Important

Applications using the Dynamic Abstract Generation API need to check out the Layout XL license. It does not require a separate license.

Related Topics

Prerequisites for Dynamic Abstract Generation

Dynamic Abstract Generation Use Model

Dynamic Abstract Generation Commands

Specifying Advanced Dynamic Abstraction Rules

© 2023 Cadence Design Systems, Inc.

Prerequisites for Dynamic Abstract Generation

Dynamic Abstract Generation uses the `Wire` constraint group that is specified in the *Wire Editing* section of the Layout Editor options form.

The prerequisites for Dynamic Abstract Generation are as follows:

- Valid vias and valid layers must be defined in the specified constraint group. If valid vias and valid layers are not present in the constraint group, a warning message is displayed in the CIW stating that the required information be added in the technology file.
- The `minSpacing` and `minWidth` layer rules must be present in the technology file. During Dynamic Abstract Generation, the constraint groups are searched for layer rules in the following order:
  - `Wire` (user-defined) constraint group
  - `LefDefaultRouteSpec` constraint group by type
  - `LefDefaultRouteSpec` constraint group by name
  - `Foundry` constraint group

  For example, if the `minSpacing` value for *metal3* is required, the `minSpacing` value is first searched for in the `Wire` constraint group. If the `minSpacing` value is found in the `Wire` constraint group, it is used. If the `minSpacing` value is not found in the `Wire` constraint group, the `minSpacing` value is searched for in the `LefDefaultRouteSpec` constraint group first by type and then by name. If the `minSpacing` value is not found in the `LefDefaultRouteSpec` constraint group too, the `Foundry` constraint group is searched for the `minSpacing` value.
- Net and pin technology purpose must be present in the technology file. During Dynamic Abstract Generation, the net and pin purpose is used for net extraction and pin creation. If the net and pin purpose is not present in the technology file, Dynamic Abstract Generation adds the layer purpose pair information in the technology file, provided the write permission is available for the technology file.
- If an `alternate foundry` constraint group (AFCG) is defined in the `.cdsenv` file, Dynamic Abstract Generator replaces the `foundry` constraint group and uses the specified AFCG instead.

  You can specify an AFCG by setting the following environment variable in your `.cdsenv` file:

  ```
  cdba.layout AlternateFoundryCG string "AFCG_name"
  ```


© 2023 Cadence Design Systems, Inc.

Related Topics

Wire Constraint Group

© 2023 Cadence Design Systems, Inc.

Dynamic Abstract Generation Use Model

In an OpenAccess design that contains Pcells, you can run Dynamic Abstract Generation manually by using the commands that can be accessed from the following three locations:

- *Tools – Dynamic Abstract Generation for Pcells*
- Navigator assistant (by using the context-sensitive menu). For example, you can use the Navigator assistant to run Dynamic Abstract Generation for Module Generator (Modgen) instances. To do this, select a Modgen instance in the Navigator assistant and select *Dynamic Abstract – Abstract Pcells* from the context-sensitive menu.
- Layout canvas (by using the context-sensitive menu)

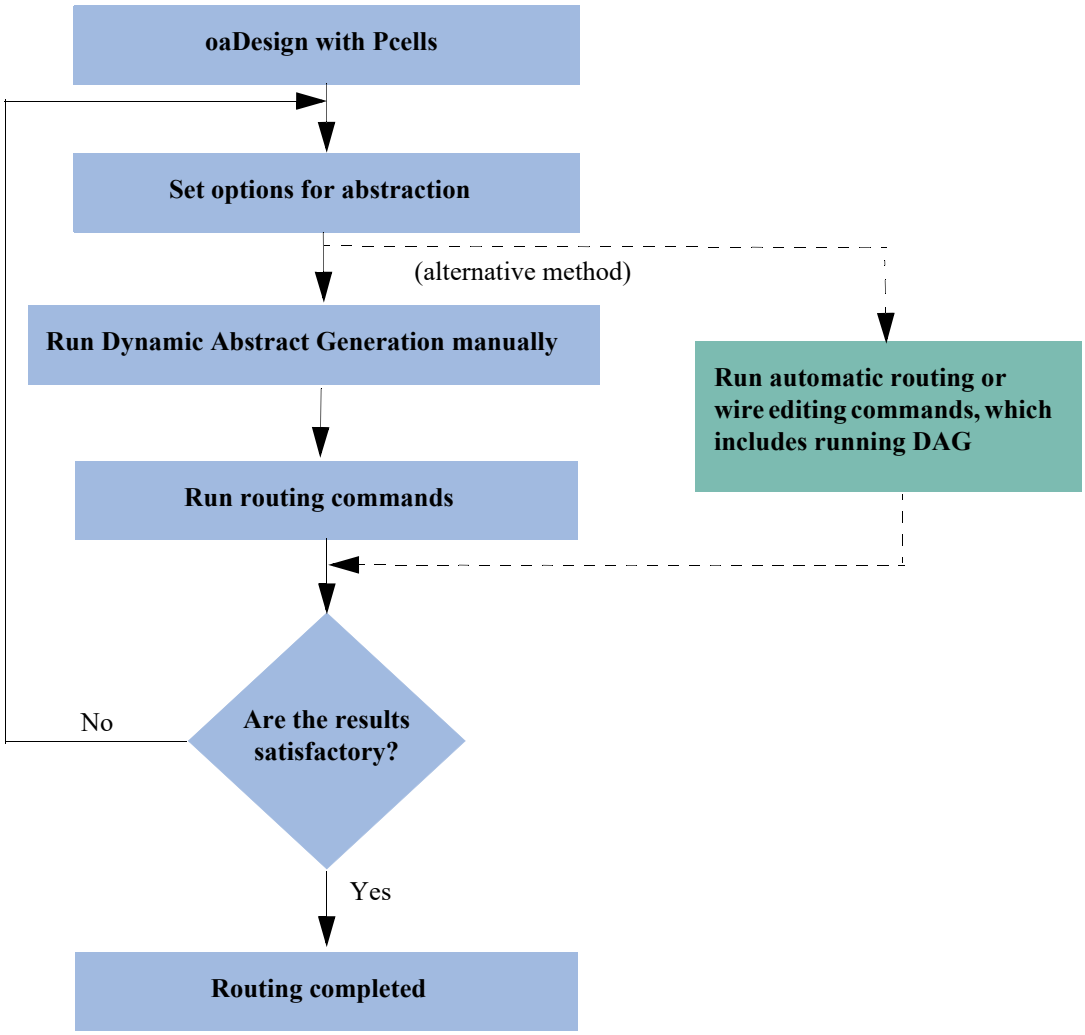
You can create abstracts for all the Pcells in the design or for only the selected Pcells, and then run the routing or wire editing commands. To create abstracts, you use either the default options or specify your own abstraction rules in the Abstract Pcells form.

When you run the automatic routing or wire editing commands, Dynamic Abstract Generation is run in the background for all the Pcell instances based on the predefined abstraction rules.

If the routing results are satisfactory, the routing process is complete. Otherwise, you can modify the abstraction rules, such as modify layer information for blockages, as required and run the process until satisfactory results are obtained.

© 2023 Cadence Design Systems, Inc.

The following flow diagram illustrates the Dynamic Abstract Generation process.



By default, all Pcells are marked for abstraction. If a Pcell is unmarked for abstraction, the Pcell needs to be marked again for abstraction.

Related Topics

Marking or Unmarking Pcells for Abstraction

Working with Module Generators

Abstracting Pcells

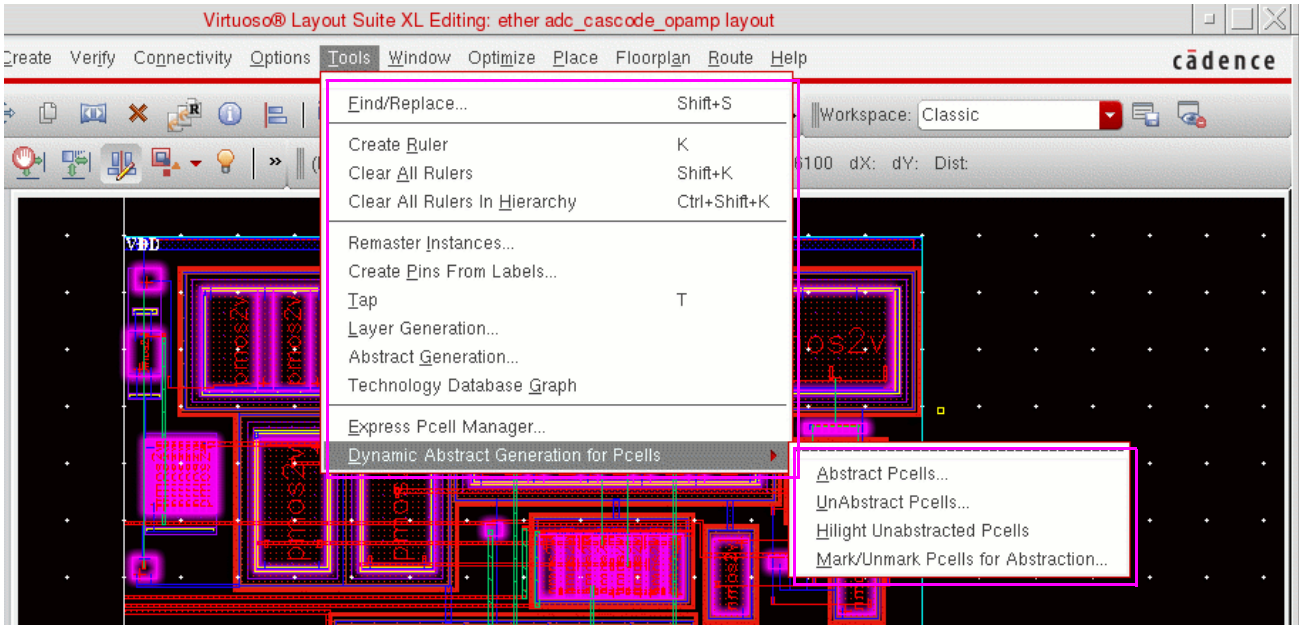
© 2023 Cadence Design Systems, Inc.

Dynamic Abstract Generation Commands

You can use the Dynamic Abstract Generation commands to abstract, unabstract, highlight, mark, and unmark Pcells.

➡ To access the commands in the layout window, choose *Tools – Dynamic Abstract Generation for Pcells*.

A submenu is displayed with the commands shown in the figure below.



Related Topics

Using Dynamic Abstract Generator

Abstracting Pcells

Dynamic Abstract Generator Context-Sensitive Menu

Highlighting Unabstracted Pcells

Marking or Unmarking Pcells for Abstraction

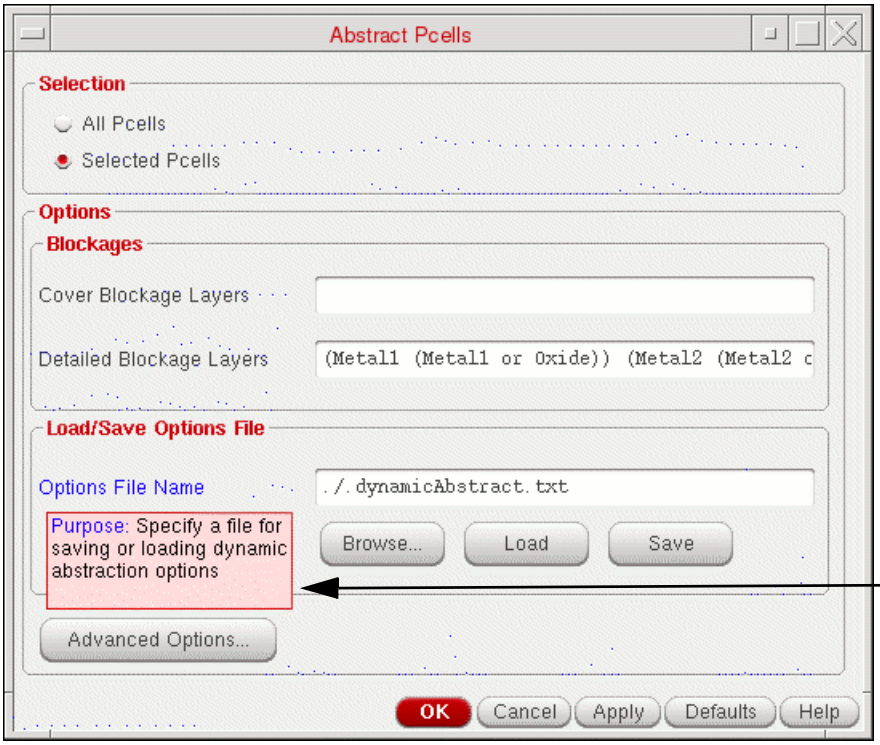
© 2023 Cadence Design Systems, Inc.

Abstracting Pcells

To generate an abstract of a Pcell in the layout window:

1. Choose *Tools – Dynamic Abstract Generation For Pcells – Abstract Pcells*.

   The Abstract Pcells form is displayed as shown in the figure below. When you point to an option in the Abstract Pcells form, an info balloon is displayed that describes the purpose of the option.



Info Balloon displaying the purpose of the option

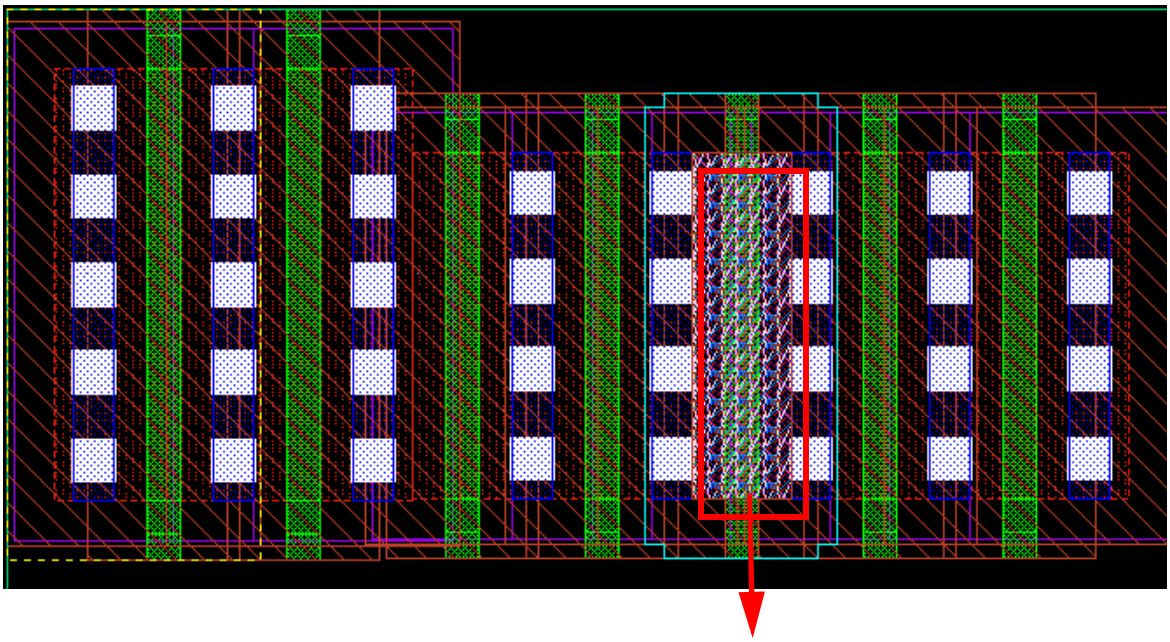
2. In the *Selection* section, choose *All Pcells* to create abstracts for all the Pcells in the design. If you do not want to create the abstracts for all the Pcells, you can select the required Pcells in the layout canvas or from the Navigator assistant and ensure that the *Selected Pcells* option is selected. After you have run the command, you can select additional Pcells for abstraction, if required, and run the command again.

   The abstracts are created for all Pcells that are selected, except the ones that are unmarked for abstraction.

3. Check whether the abstracts were created successfully using one of the following methods:

© 2023 Cadence Design Systems, Inc.

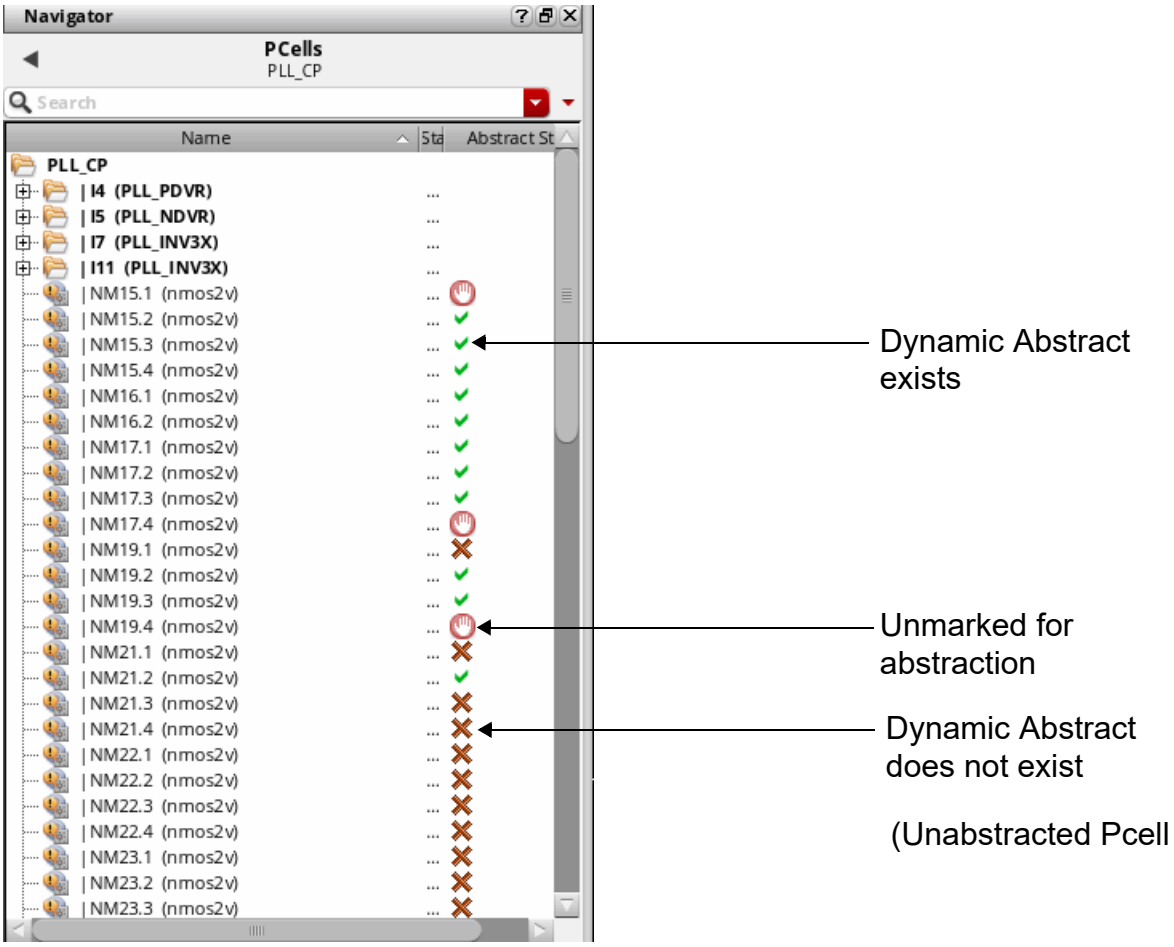
- View the blockages created at level-1in the layout canvas, as shown in the figure below.



Blockages visible for an abstracted Pcell

© 2023 Cadence Design Systems, Inc.

- View the status icons associated with the various Pcells in the Navigator assistant.



4. In *Options – Blockages* section, you can specify the settings for blockage generation and for loading or saving an options file.
5. Use *Cover Blockage Layer*s to specify layers and choose *Detailed Blockage Layers* to specify a layer-geometry specification.
6. Select *Load/Save Options File* to load or save the options file. You can use the following methods to load and save an options file:
   - Save the abstraction rules specified in the Abstract Pcells form to an options file.
     - Click *Save* to save the abstraction rules to an options file.
     - Specify the name of the options file to which you want to save the abstraction rules in the *Options File Name* field. You can also load an existing options file, edit it, and save the changes.

© 2023 Cadence Design Systems, Inc.

- Load the various rules contained in an options file to the Abstract Pcells form. To load the abstraction rules contained in an options file, select the required options file and click *Load*.

Related Topics

Abstract Pcells Form

Dynamic Abstract Generation Use Model

Loading and Saving an Options File for Dynamic Abstract Generation

© 2023 Cadence Design Systems, Inc.

Loading and Saving an Options File for Dynamic Abstract Generation

You can load or save the options file, which is an editable text file that contains the abstraction rules used by Dynamic Abstract Generation to create abstracts.

To set the `dynamicAGsFile` environment variable to specify the options file that you want to load:

1. Choose *Tools – Dynamic Abstract Generation for Pcells – Abstract Pcell*s.

   By default, the `dynamicAGsFile` environment variable loads the options file named `dynamicAbstract.option` when the Abstract Pcells form is opened for the first time in a Virtuoso session.

2. You can then change the values of the fields and click *OK*.

   The next time when you open the Abstract Pcells form in the same Virtuoso session, the `dynamicAbstract.option` file is not loaded and the modified values, if any, are retained.

You can either read the `dynamicAGsFile` environment variable from the `.cdsenv` file or set it in the CIW. To set environment variable, in the CIW, type:

```
envGetVal ("layout XL" "dynamicAGsFile" 'string "dynamicAGsFile")
```

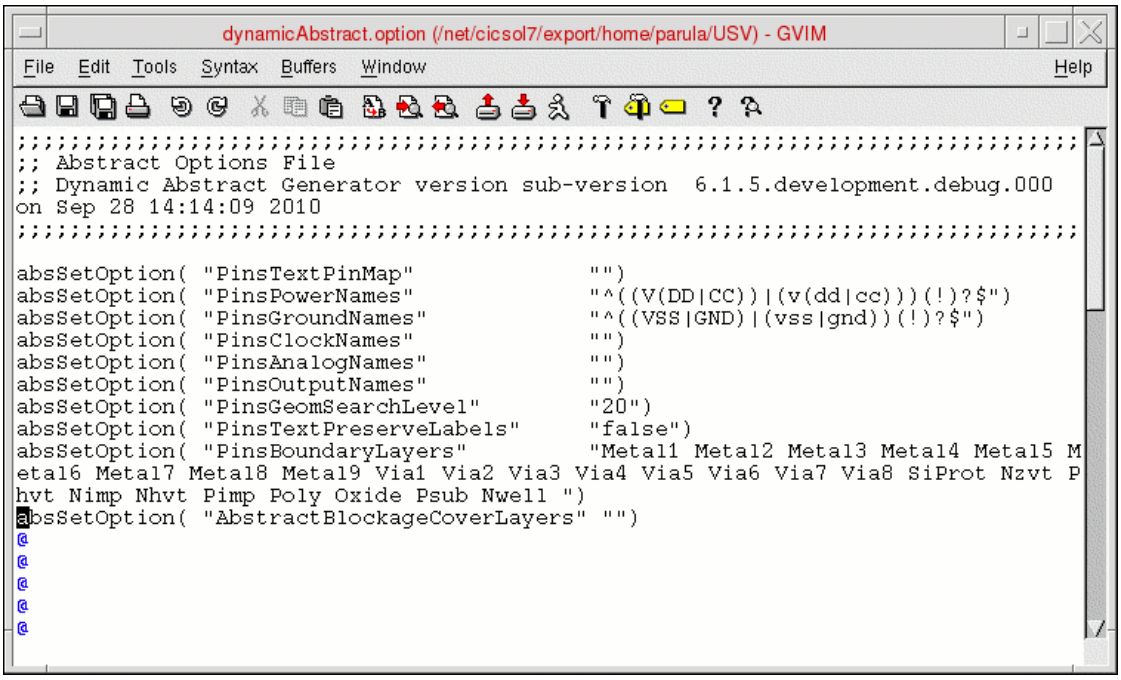
Here, the `dynamicAGsFile` is the name of the options file that is to be loaded.

You can use the following methods to load and save an options file:

- Save the abstraction rules specified in the Abstract Pcells form to an options file.To save the abstraction rules to an options file, click *Save*. Specify the name of the options file to which you want to save the abstraction rules in the *options File Name* field. You can also load an existing options file, edit it, and save the changes.
- Load the various rules contained in an options file to the Abstract Pcells form. To load the abstraction rules contained in an options file, select the required options file and click *Load*.
- Directly edit an existing options file in a text editor and save the changes.

© 2023 Cadence Design Systems, Inc.

The following figure shows a sample options file.



Related Topics

Abstract Pcells Form

© 2023 Cadence Design Systems, Inc.

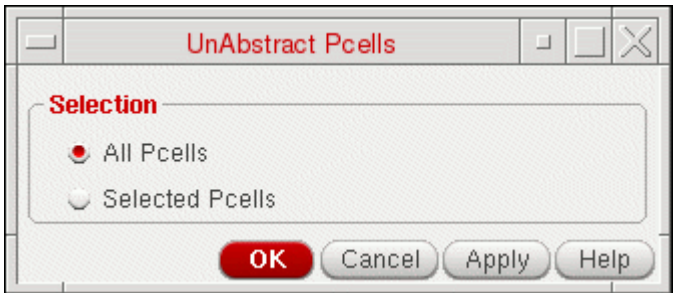
Unabstracting Pcells

Unabstracting a Pcell involves deleting the abstract data that exists for a Pcell. If an abstract data that you created for a Pcell does not meet your requirements and you want to create the abstract data again based on a new set of abstraction options, you need to first unabstract a Pcell and then run Dynamic Abstract Generation again.

To unabstract a Pcell, in the Layout window,

1. Choose *Tools – Dynamic Abstract Generation For Pcells – UnAbstract Pcells*.

   The UnAbstract Pcells form is displayed.



2. Select *All Pcells* or *Selected Pcells* to specify if you want to unabstract all Pcells or the selected Pcells.
3. Click *OK*.

Related Topics

UnAbstract Pcells Form

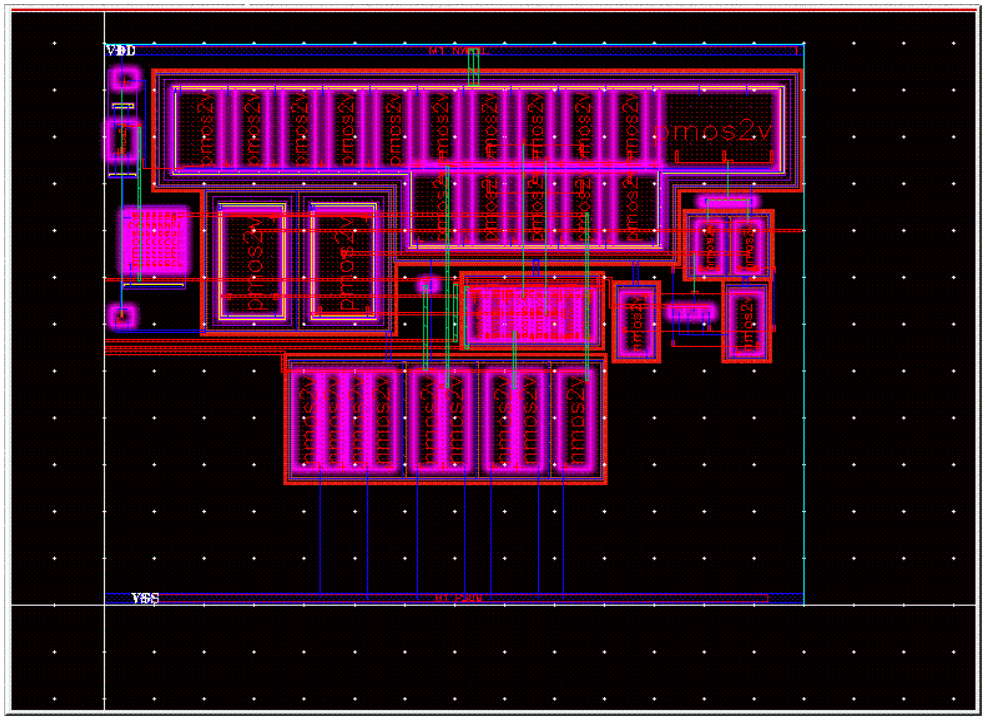
© 2023 Cadence Design Systems, Inc.

Highlighting Unabstracted Pcells

To highlight the Pcells that are marked for abstraction but have not yet been abstracted:

➡ In the layout window, choose *Tools – Dynamic Abstract Generation From Pcells – Highlight Unabstracted Pcells*.

A highlight is displayed around each Pcell that has not been abstracted, as shown below.



If you click anywhere in the layout canvas, the highlights around the unabstracted Pcells are removed.

Related Topics

Unabstracting Pcells

UnAbstract Pcells Form

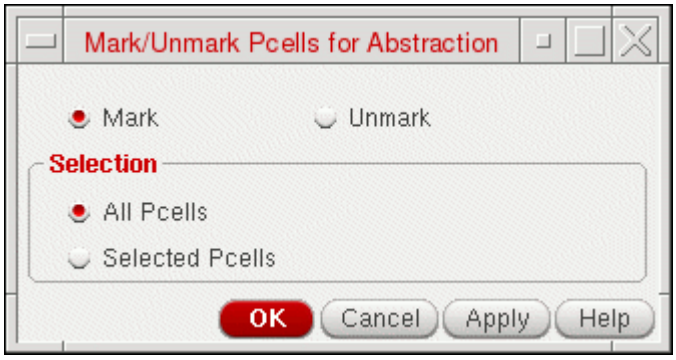
© 2023 Cadence Design Systems, Inc.

Marking or Unmarking Pcells for Abstraction

To mark or unmark Pcells for abstraction.

1. Choose *Tools – Dynamic Abstract Generation for Pcells – Mark/Unmark Pcells for Abstraction*.

   The Mark/Unmark Pcells for Abstraction form is displayed.

   You can either mark or unmark the Pcells for abstraction.

2. Select *Mark* to mark a Pcell for abstraction. This is also the default option.
3. Select *Unmark* to unmark a Pcell.
4. Select *All Pcells* or *Selected Pcells* to specify if you want to unabstract all Pcells or the selected Pcells.

As a result, the Pcell is ignored for the abstraction process. By default, all the Pcells are marked for abstraction.

Related Topics

Mark/Unmark Pcells for Abstraction Form

UnAbstract Pcells Form

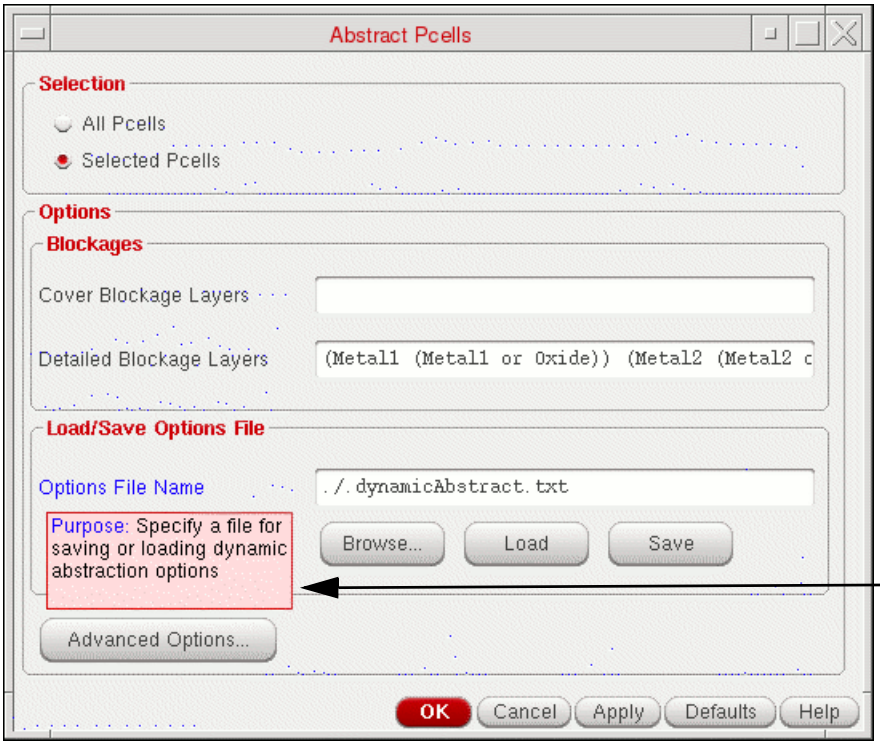
© 2023 Cadence Design Systems, Inc.

Specifying Advanced Dynamic Abstraction Rules

To specify the advanced abstraction rules in the layout window:

1. Choose *Tools – Dynamic Abstract Generation For Pcells – Abstract Pcells*.

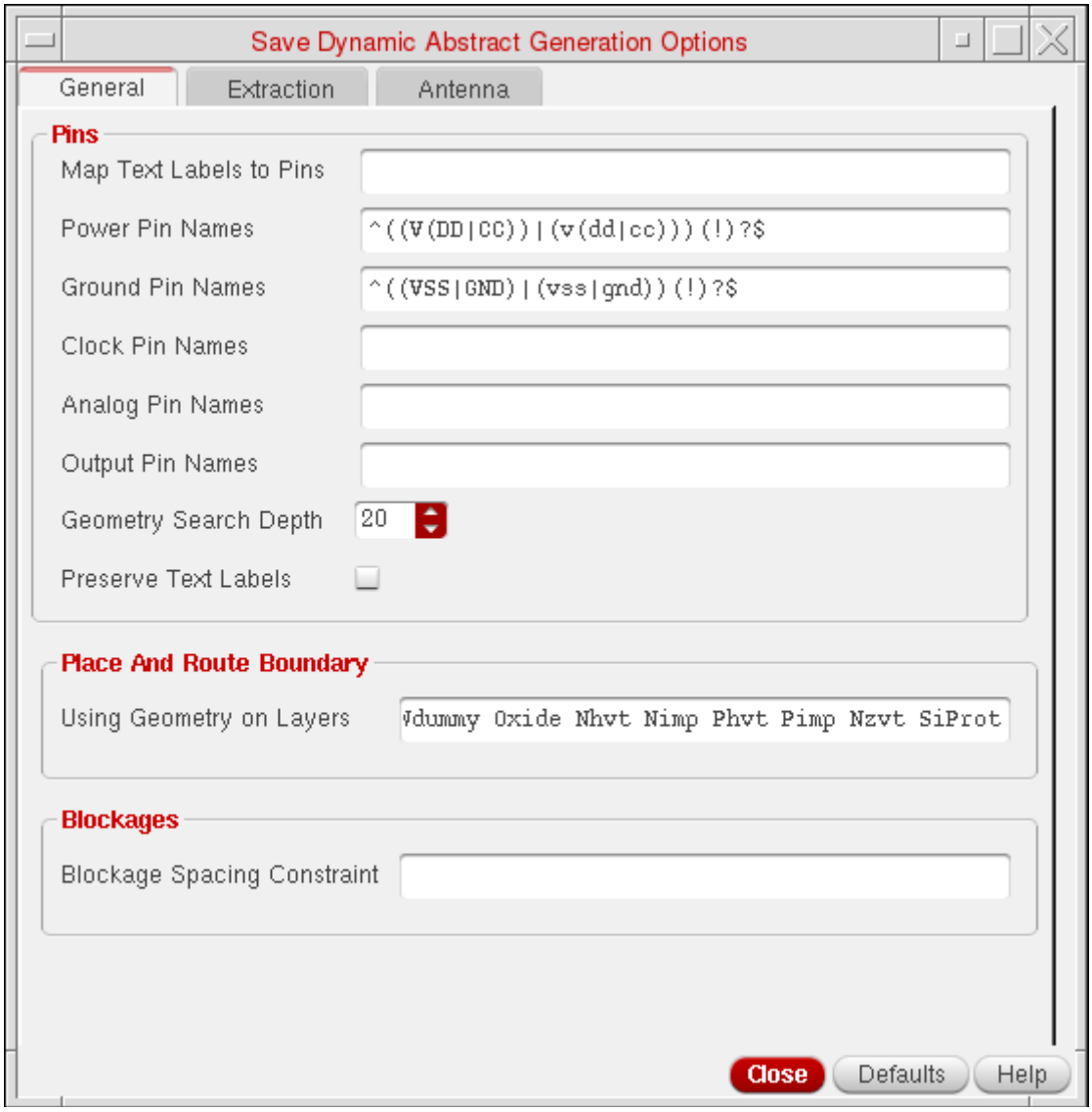
   The Abstract Pcells form is displayed as shown in the figure below.



2. Click *Advanced Options*.

© 2023 Cadence Design Systems, Inc.

The Save Dynamic Abstract Generation options form is displayed as shown in the following figure. The form has three tabs: *General*, *Extraction*, and *Antenna*.



Related Topics

Mark/Unmark Pcells for Abstraction Form

Specifying General Dynamic Abstraction Options

Specifying Extraction Options for Dynamic Abstraction

Specifying Antenna Options for Dynamic Abstraction

© 2023 Cadence Design Systems, Inc.

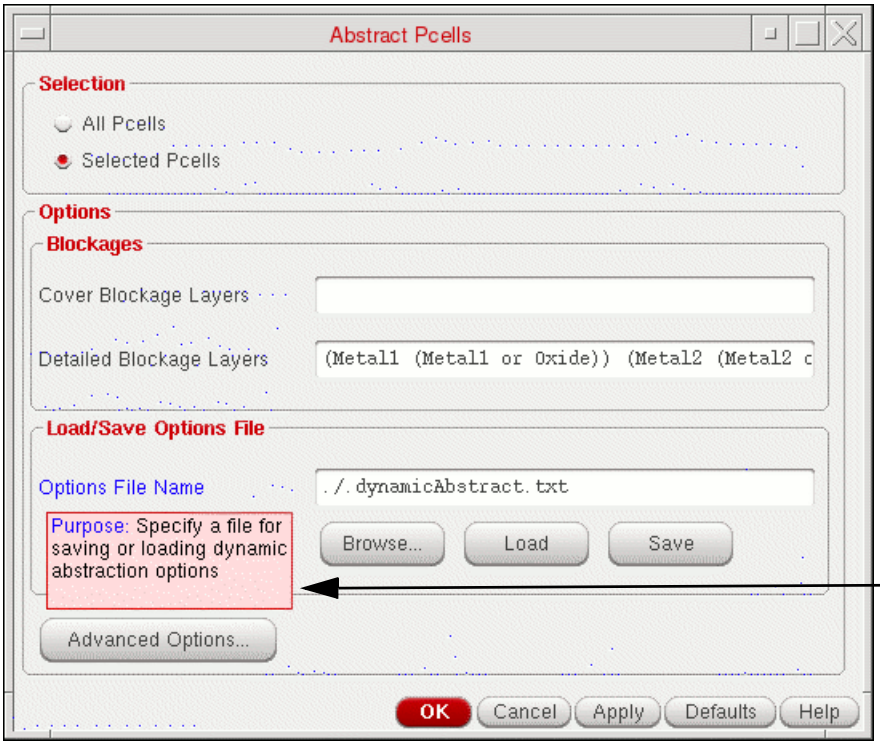
Specifying General Dynamic Abstraction Options

You can specify pin, PR boundary and blockages settings in the *General* tab of the Save Dynamic Abstract Generation Options form.

To use the general settings in the Save Dynamic Abstract Generation Options form:

1. Choose *Tools – Dynamic Abstract Generation For Pcells – Abstract Pcells*.

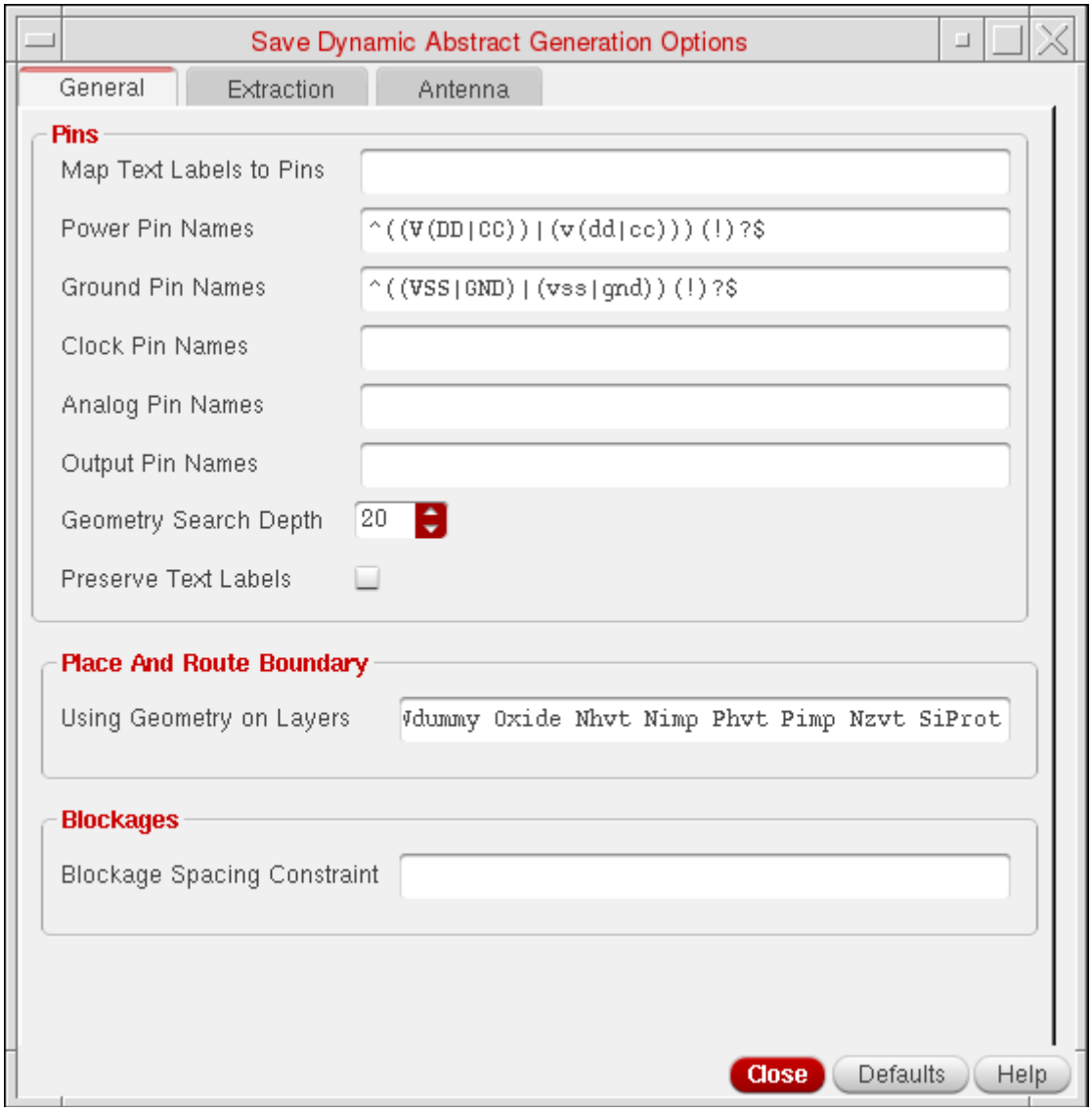
   The Abstract Pcells form is displayed.



2. Click *Advanced Options*.

© 2023 Cadence Design Systems, Inc.

The Save Dynamic Abstract Generation options form is displayed as shown in the following figure. The form has three tabs: General, Extraction, and Antenna tab.



3. In the *Pins* section, you can specify the pin settings.

   - Choose *Map Text Labels to Pins* to specify the text labels that you want to map from the layout view to be created as pins in the layout dual view. This field is case-sensitive.
   - In the *Power Pin Names*, *Ground Pin Names*, *Clock Pin Names*, *Analog Pin Names,* and *Output Pin Names* fields, specify a list of names for power pin, ground pin, clock pin, analog pin, and output pin, respectively. Specifying the pin names enables you to distinguish the pin types in the design.

© 2023 Cadence Design Systems, Inc.

Note: By default, the *Clock Pin Names*, *Analog Pin Names*, and *Output Pin Names* fields are empty.

- In the *Geometry Search Depth* box, specify the hierarchy depth.
- Select the *Preserve text labels* check box to retain the text labels from the layout view in the layout dual view.

4. In the *Using geometry on layers* field, specify a list of layers or layer purpose pairs to be used to define the prBoundary.
5. In the *Blockage Spacing Constraint* field, specify a list of layer purpose pairs on which you want to define a spacing constraint along with the spacing value.
6. Click *Close* after setting the required fields.

Related Topics

Save Dynamic Abstract Generation Options Form

© 2023 Cadence Design Systems, Inc.

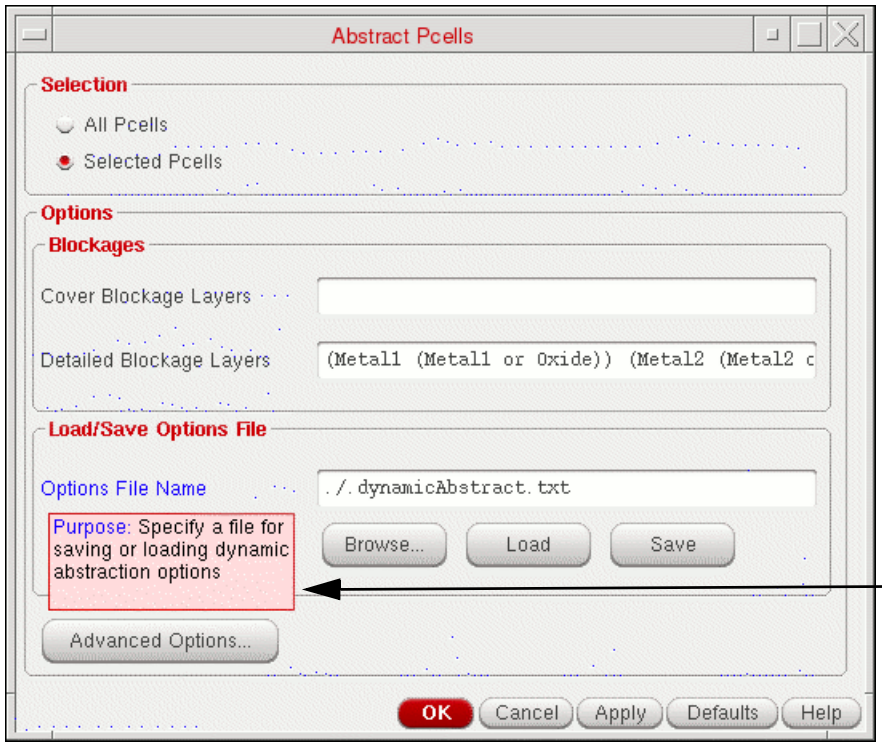
Specifying Extraction Options for Dynamic Abstraction

You can specify the extraction information for signal and power nets in the *Extraction* tab of the Save Dynamic Abstract Generation Options form.

To use the settings in the *Extraction* tab of the Save Dynamic Abstract Generation Options form:

1. Choose *Tools – Dynamic Abstract Generation For Pcells – Abstract Pcells*.

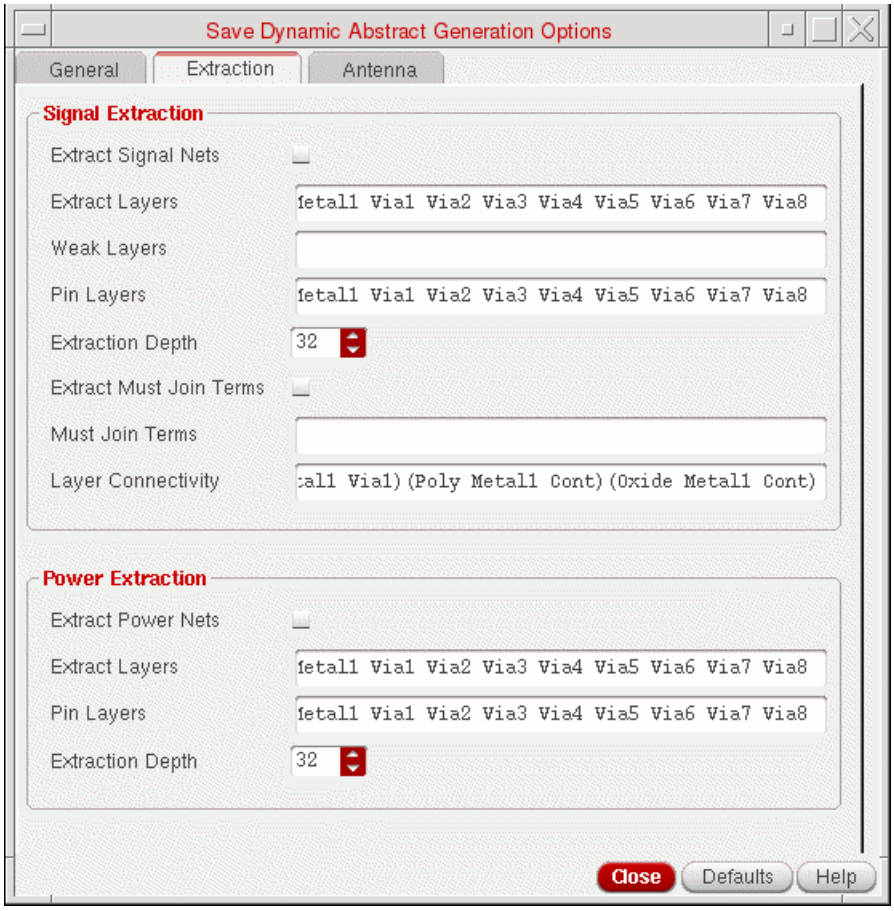
   The Abstract Pcells form is displayed as shown in the figure below.



2. Click *Advanced Options*.

© 2023 Cadence Design Systems, Inc.

The Save Dynamic Abstract Generation options form is displayed as shown in the following figure. The form has three tabs: *General*, *Extraction*, and *Antenna* tab.



3. In the *Signal Extraction* section, you can specify the following options for extracting signal nets.

   - Select *Extract Signal Nets* to run connectivity extraction on signal nets. By default, the *Extract Signal Net* check box is deselected.
   - Use *Extract Layers* to specify the layers or layer purpose pairs on which signal net extraction is to be performed.
   - Use *Weak Layers* field to specify the layers that you want to be considered as weak layers. The weakly connected pins will be created for the layers specified in the *Weak Layers* field. By default, the *Weak Layers* field is empty.
   - Use *Pin Layers* field to specify the layers on which you want the pins to be created. This can be a subset of layers that are specified in the *Extract Layers* field.

© 2023 Cadence Design Systems, Inc.

- Select *Extraction Depth* field to specify how many levels down the instance hierarchy the Dynamic Abstract Generation process must search for the shapes to be extracted.
- Select *Extract Must Join Terms* to establish a must-join relationship between all disjoint groups of geometries extracted from separate pins on the same net.
- Select *Must Join Term* to specify a list of terminal names, each separated by a space.
- Select *Layer connectivity* to specify the connectivity between layers. This enables you to specify the layer connectivity without having to add via definitions in the technology file.

4. In the *Power Extraction* section, you can specify the following options for extracting power nets.

- Select *Extract Power Nets* to run connectivity extraction for power nets. If this check box is selected, all the pin shapes created during the Pins step are extracted. By default, the *Extract Power Nets* check box is deselected.
- Use *Extract Layers* to specify the layers or layer purpose pairs through which you want the power nets to be extracted.
- Use *Pin Layers* to specify the layers on which you want the pins to be created. Creating pins on a layer helps to determine the shapes found by the extractor that are converted into pins.
- Select *Extraction Depth* to specify how many levels down the instance hierarchy the Dynamic Abstract Generation process must search for the shapes to be extracted. The default value is `32`, which is the maximum number of levels in a design hierarchy stored in the database.

5. Click *Close* after setting the required fields.

© 2023 Cadence Design Systems, Inc.

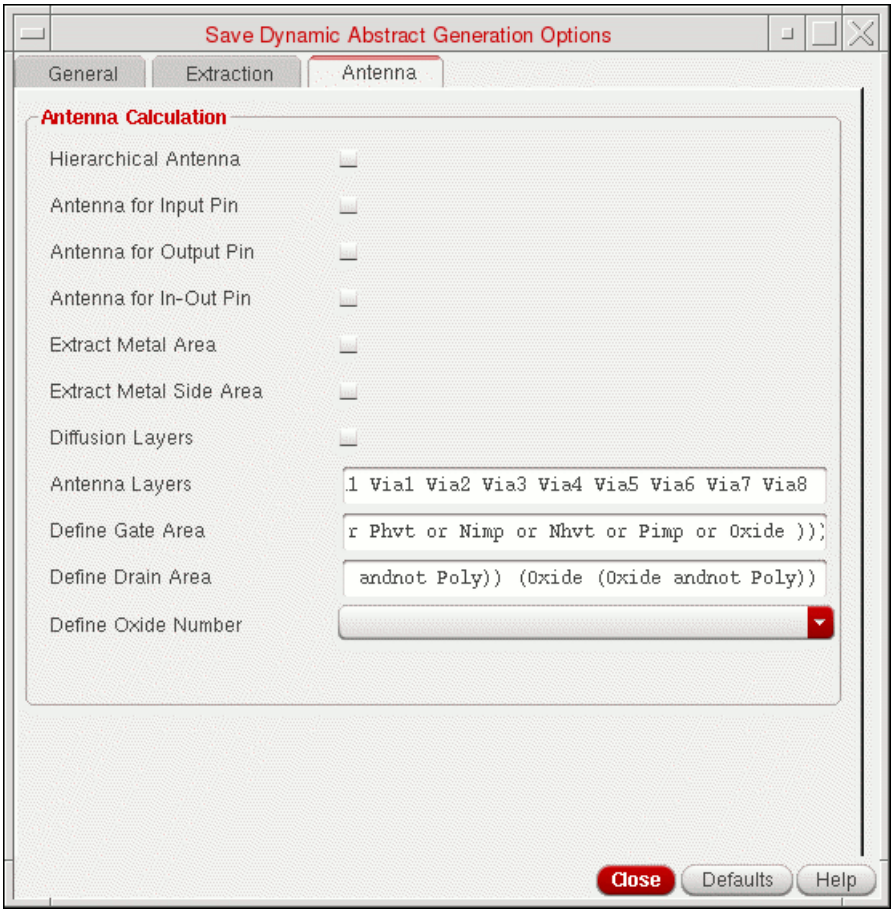
Related Topics

Save Dynamic Abstract Generation Options Form

© 2023 Cadence Design Systems, Inc.

Specifying Antenna Options for Dynamic Abstraction

You can specify antenna options for dynamic Abstraction using the Save Dynamic Abstract Generation Options form. By using the options on the *Antenna* tab, you can create process antenna data for Pcells. By default, all the check boxes on this tab are deselected.



1. In the *Antenna Calculation* section, use the following options to computes the process antenna data for a given layer.
   - ❑ *Hierarchical Antenna*
   - ❑ *Antenna for Input Pin*
   - ❑ *Antenna for Output Pin*
   - ❑ *Antenna for In-Out Pin*
   - ❑ *Extract Metal Area*
   - ❑ *Extract Metal Side Area*

© 2023 Cadence Design Systems, Inc.

2. Select the *Diffusion Layers* check box to extract diffusion layers specified in the technology file.
3. In the *Antenna Layers* field, specify the layers for which you want to calculate their process antenna data.
4. In the *Define Gate Area* and *Define Drain Area* fields, specify the layer-geometry specification for defining the gate and drain geometry regions, respectively. An example that defines the gate area region is as follows:

   ```
   (poly and fiff)
   ```

   An example that defines the drain region area is as follows:

   ```
   (diff andnot poly)
   ```
5. From the *Define Oxide Number* drop-down list, select the oxide model to be extracted for a particular diffusion layer. The list contains four oxide types: `Oxide1`, `Oxide2`, `Oxide3`, and `Oxide4`. You can apply a unique oxide model for up to four diffusion layers.
6. Click *Close* after setting the required fields.

Related Topics

Save Dynamic Abstract Generation Options Form

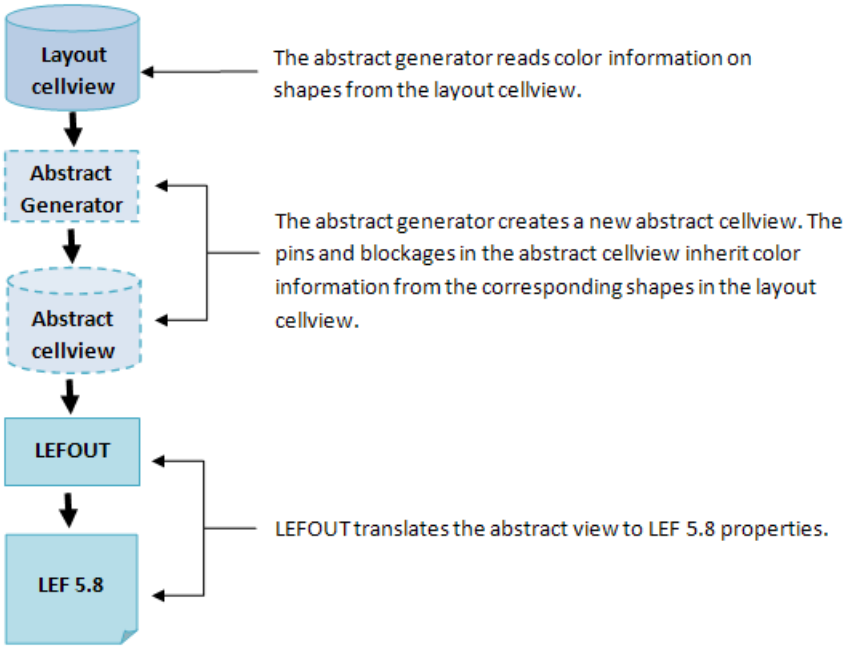
© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

8

Multi-Patterning Technology Support in Abstract Generator

Virtuoso® Abstract Generator supports Multi-Patterning Technology (MPT). To handle color, the abstract representation includes the color of pins and obstructions. The following diagram depicts the process of generating a color-annotated abstract cellview from a color-annotated layout cellview and exporting the abstract cellview to LEF 5.8.



© 2023 Cadence Design Systems, Inc.

Pin and obstruction colors are represented in LEF 5.8 as:

```
    PORT
      LAYER MET1 ;
        POLYGON MASK1 1.108 0.432 0.352 0.432 0.352 0.4 1.076 0.4
1.076 0.112 0.344 0.112 0.344 0.08 1.108 0.08 ;
      LAYER MET0 ;
        RECT MASK1 1.038 0.084 1.062 0.166 ;
        RECT MASK1 1.038 0.346 1.062 0.428 ;
        RECT MASK1 0.87 0.084 0.894 0.166 ;
        RECT MASK1 0.87 0.346 0.894 0.428 ;
        RECT MASK1 0.702 0.084 0.726 0.166 ;
        RECT MASK1 0.702 0.346 0.726 0.428 ;
        RECT MASK1 0.534 0.084 0.558 0.166 ;
        RECT MASK1 0.534 0.346 0.558 0.428 ;
        RECT MASK1 0.366 0.084 0.39 0.166 ;
        RECT MASK1 0.366 0.346 0.39 0.428 ;
      LAYER VIA0 ;
        RECT 1.034 0.08 1.066 0.112 ;
        RECT 1.034 0.4 1.066 0.432 ;
        RECT 0.866 0.08 0.898 0.112 ;
        RECT 0.866 0.4 0.898 0.432 ;
        RECT 0.698 0.08 0.73 0.112 ;
        RECT 0.698 0.4 0.73 0.432 ;
        RECT 0.53 0.08 0.562 0.112 ;
        RECT 0.53 0.4 0.562 0.432 ;
        RECT 0.362 0.08 0.394 0.112 ;
        RECT 0.362 0.4 0.394 0.432 ;
    END
  END y
  PIN vdd
    DIRECTION INOUT ;
    USE POWER ;
    SHAPE ABUTMENT ;
    PORT
      LAYER MET2 ;
        RECT 0 0.496 1.176 0.528 ;
    END
  END vdd
  PIN vss
    DIRECTION INOUT ;
    USE GROUND ;
    SHAPE ABUTMENT ;
    PORT
      LAYER MET2 ;
        RECT 0 -0.016 1.176 0.016 ;
    END
  END vss
  OBS
    LAYER MET1 ;
      RECT MASK2 0.03 0.08 0.268 0.112 ;
      RECT MASK2 0.03 0.08 0.062 0.16 ;
      RECT MASK2 0.236 0.24 0.46 0.272 ;
      RECT MASK2 0.03 0.352 0.062 0.432 ;
      RECT MASK2 0.236 0.08 0.268 0.432 ;
      RECT MASK2 0.03 0.4 0.268 0.432 ;
      RECT MASK1 0.134 0.176 0.194 0.336 ;
      RECT MASK1 0.344 0.08 1.108 0.112 ;
      RECT MASK1 1.076 0.08 1.108 0.432 ;
```

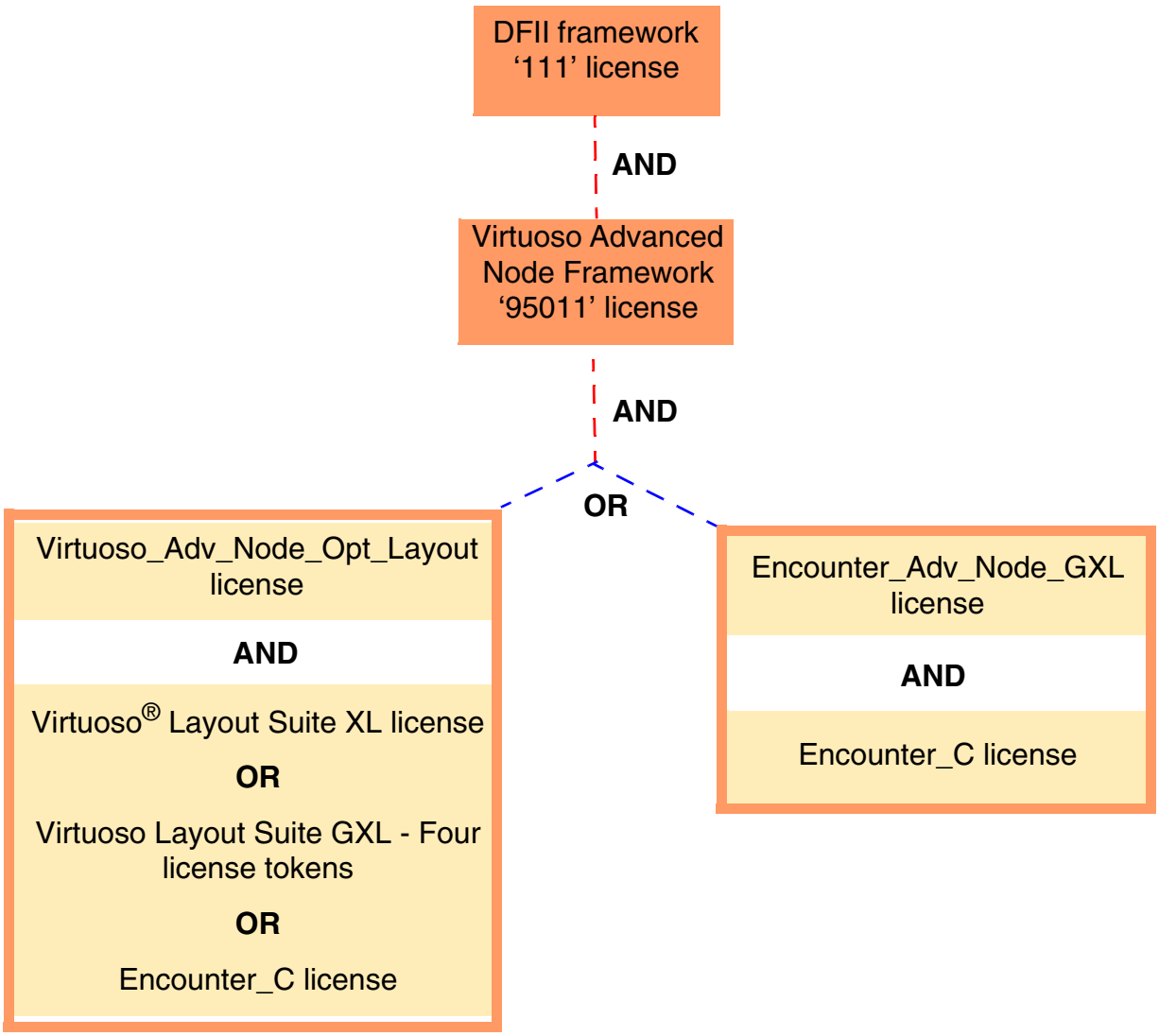
Abstracts do not include cluster information because routing applications such as Innovus may later shift the colors on a per-instance basis to avoid color conflict.

© 2023 Cadence Design Systems, Inc.

Prerequisites for Multi-Patterning Technology Support

The section below details the prerequisites for Virtuoso multi-patterning technology:

- The technology file must include the number mask.
- The following figure describes the licensing requirements for Abstract Generator when using Virtuoso Multi-Patterning Technology:



Related Topics

Color Representation for Pins

© 2023 Cadence Design Systems, Inc.

Supported Blockage Colors in Multi-Patterning Technology

Support for Trim Metal

Importing GDSII Information in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Color Representation for Pins

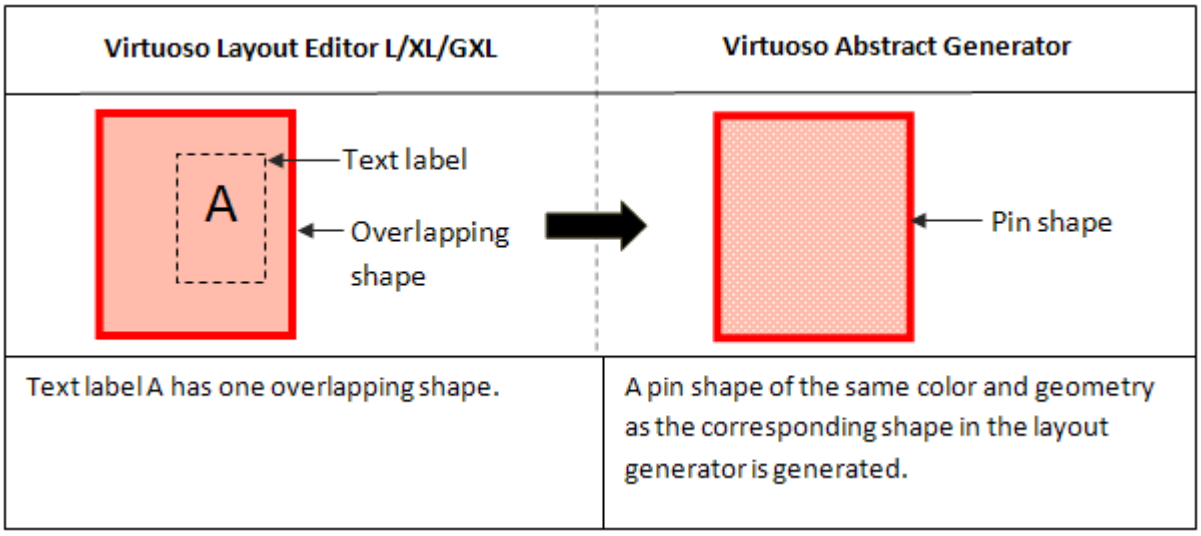
This section provides information about the implementation of MPT in the following abstract generation steps.

- Pins Step
- Extract Step
- Abstract Step

Pins Step

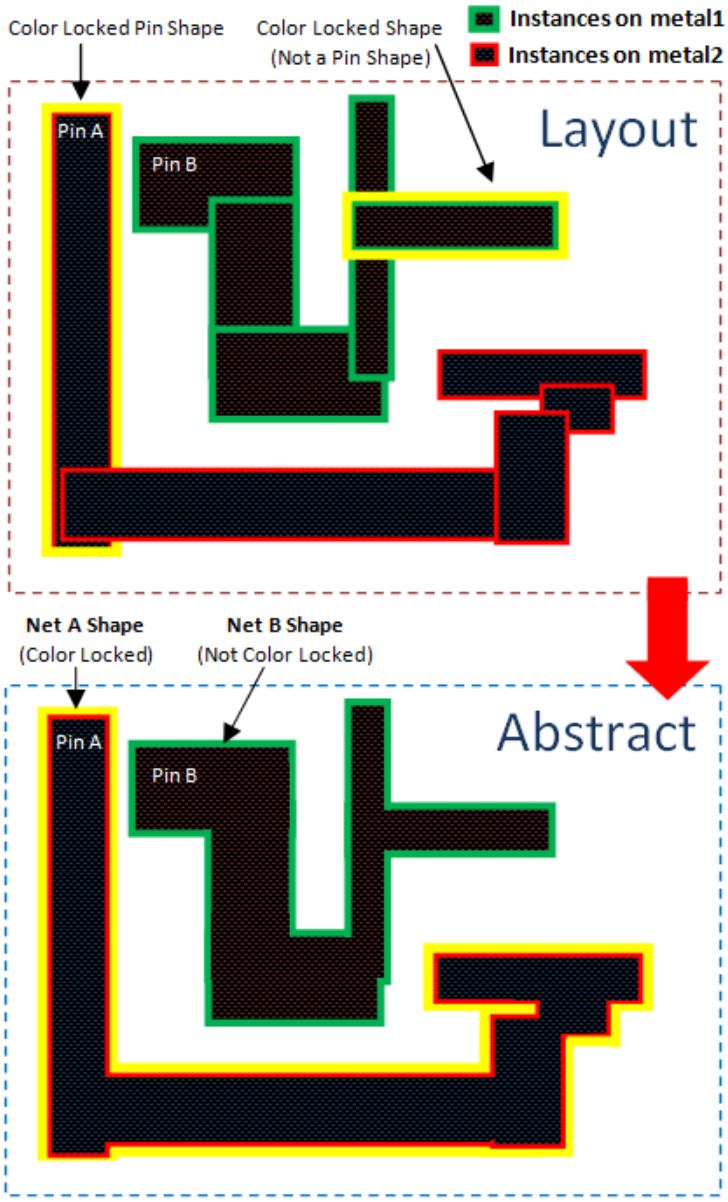
Generating pins is the first step of the abstract generation process. Typically, in this step, Abstract Generator derives pin shapes from the text labels in the layout view and places them at the locations of the corresponding text labels.

To support MPT, in the layout editor, shapes are highlighted in different colors to represent different masks. During the Pins step, when searching for pin shapes and chasing shapes, Abstract Generator recognizes the effective color of shapes in the layout editor. Consequently, the pins that are generated inherit the colors of the shapes from which they are derived.



© 2023 Cadence Design Systems, Inc.

The color lock attribute of pins is forward-propagated to the abstract view. Consider a color locked pin in the layout design. In the abstract view, the color lock attribute is applied to all the pins in the layer in which the pin is located.

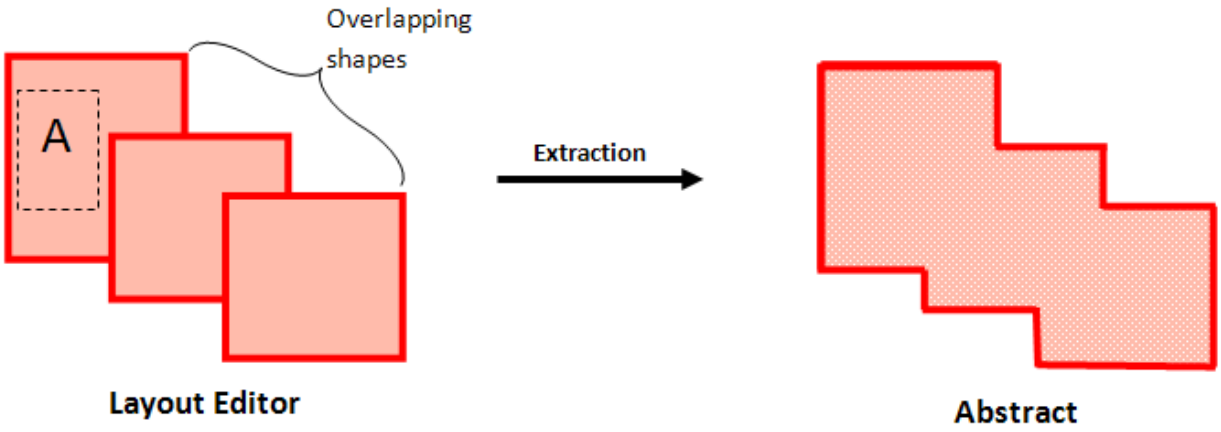


Extract Step

The Extract step is the second stage of abstract generation. This step traces the connectivity between shapes and terminals, starting at the pin shapes created in the Pins step.

© 2023 Cadence Design Systems, Inc.

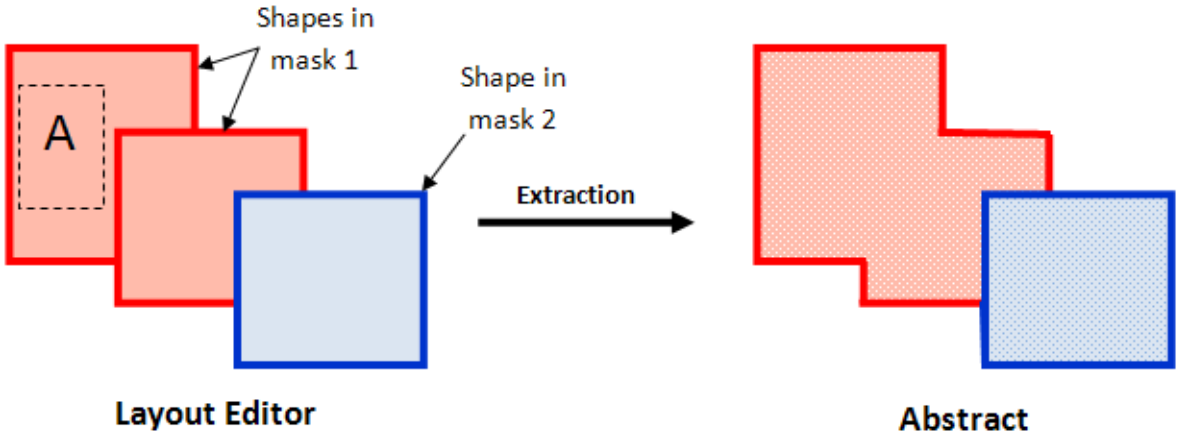
To support MPT, during extraction, overlapping pins that lie on the same layer and on the same mask on the layout are merged in the abstract view. A single shape is generated by Abstract Generator.



Overlapping pins that are located on different masks on the layout are not merged in the abstract. Different pin shapes corresponding to the different masks are generated.

Situation 1:

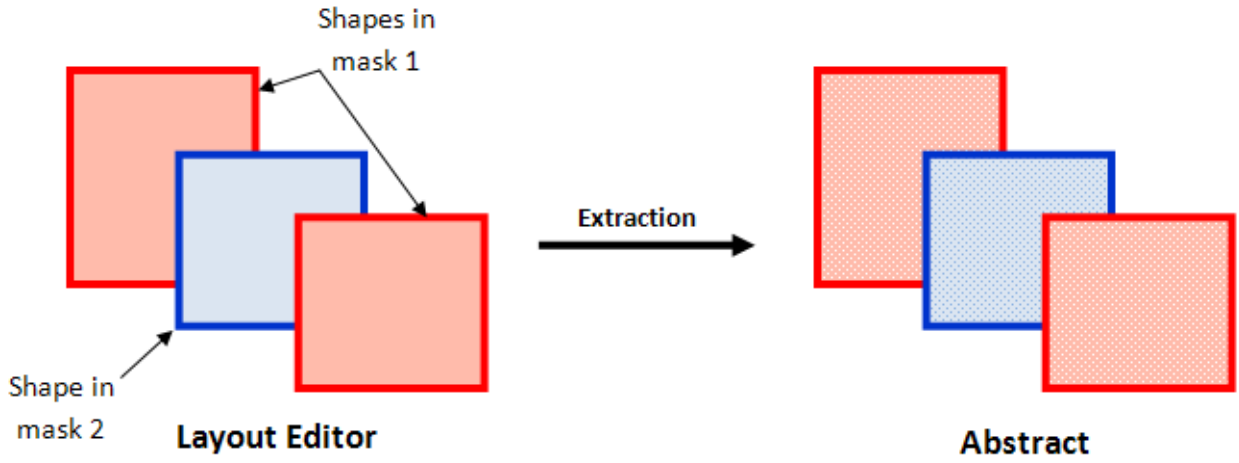
The layout editor has two overlapping shapes on Mask 1 and a third shape is on Mask 2. Notice that during extraction, overlapping shapes on the same mask are merged, whereas a separate pin shape is generated for the pin that lies on a different mask.



© 2023 Cadence Design Systems, Inc.

Situation 2:

In the layout editor, overlapping shapes are on different masks. During extraction, pin merging does not happen. Separate pin shapes for each pin are generated in the abstract.



Abstract Step

The *Abstract* step is the third step of the abstract generation process. In this step, you can use the options in the Adjust tab to adjust pin shapes for creating the final required pin shapes. These pin shapes are then fractured into rectangles.

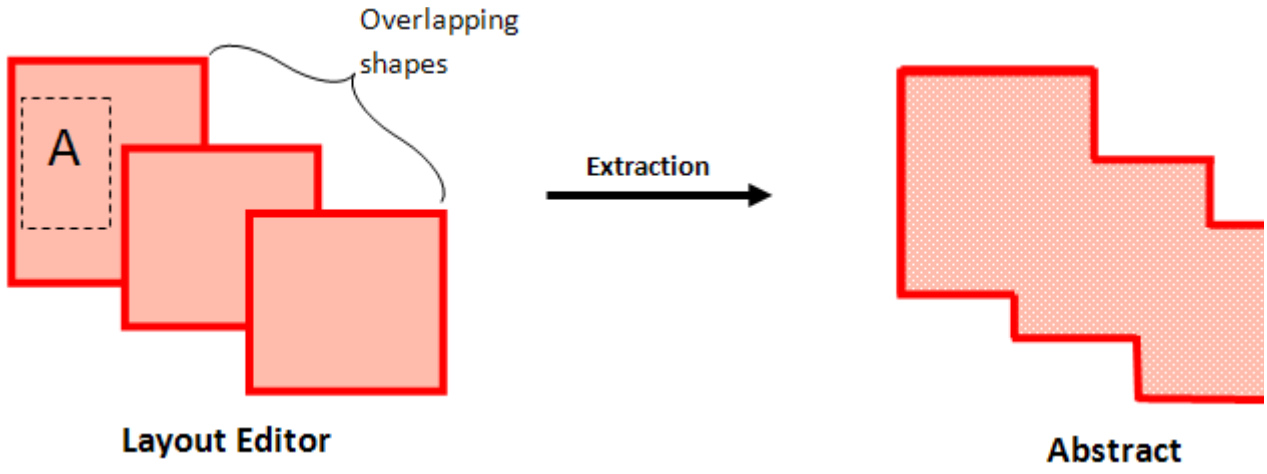
MPT support has been extended to the options on the Adjust tab of the Abstract step. Options for performing the following tasks now recognize the colors of the original shapes in the layout editor:

- Creation of square boundary pins for signal and power nets
- Creation of ring pins for power nets
- Creation of follow ring pins for power nets
- Grouping the power geometries
- Setting the options for CLASS CORE ports
- Creating CLASS BUMP ports
- Specifying power rail widths, offsets, and shapes
- Assigning layers for pin geometry
- Fracturing pins

© 2023 Cadence Design Systems, Inc.

Pin shapes created using the above options inherit colors from the original shapes. For example, two pin shapes created by fracturing a single rectilinear shape will inherit the color of the original rectilinear shape.

Consider two pin shapes, A and B located on mask 1 and mask 2, respectively. Pin A is rectangular, whereas pin B is polygonal in shape.



During the abstract step, the rectangular pin shape is not fractured, whereas the polygonal pin shape is fractured into smaller rectangles. Thus, pin B is fractured into two rectangles, which remain on mask 2.

Related Topics

Abstract Step in Standalone Abstract Generation

Extract Step in Standalone Abstract Generation

Multi-Patterning Support in Virtuoso

Pins Step in Standalone Abstract Generation

© 2023 Cadence Design Systems, Inc.

Color Representation for Blockages

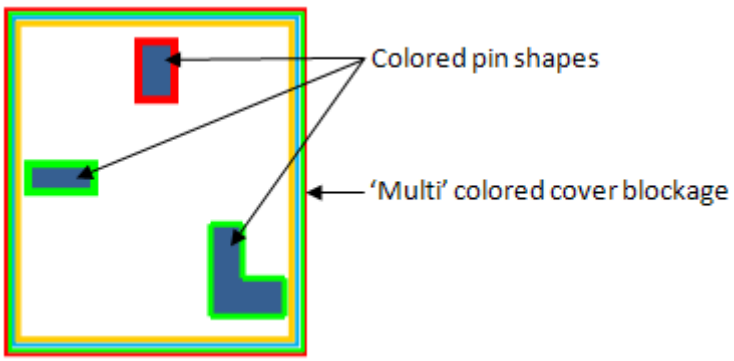
Similar to shapes, blockages can be assigned a specific or no color. Blockages with specific color can be interpreted the same way as shapes with specific colors. Blockages at the same level of hierarchy with a particular color in a layout view should be at same-color spacing distance from shapes of the same color.

Blockage colors depend on the type of blockage.

- Cover Blockage
- Shrink Blockage
- Detailed Blockage

Cover Blockage

A cover blockage covers an entire block for each metal layer used by the block. A cover blockage may have several underlying shapes of different colors. Therefore, cover blockages are assigned the ‘multi’ color option.

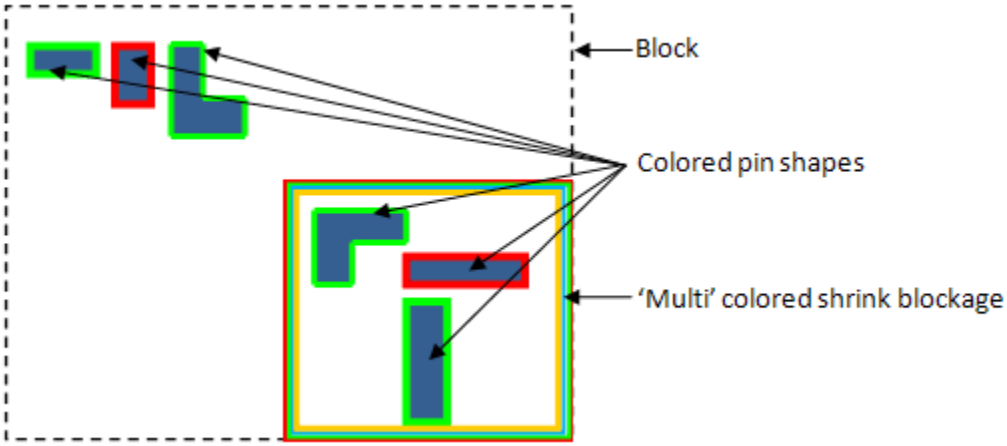


Shrink Blockage

A shrink blockage covers a set of shapes within a block. There can be multiple shrink blockages within a single block. Similar to cover blockages, shrink blockages can have

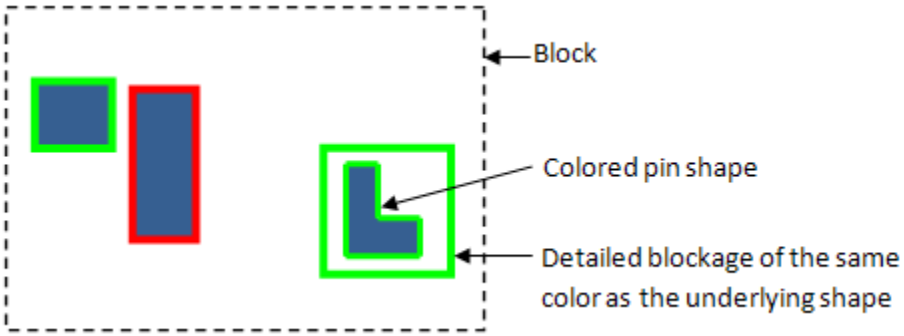
© 2023 Cadence Design Systems, Inc.

multiple underlying shapes of various colors. Therefore, shrink blockages are also assigned the ‘multi’ color option.



Detailed Blockage

Detailed blockages cover individual geometries within blocks. This type of blockage is used for standard cells and possibly for higher-level routing layers in blocks. Unlike cover and shrink blockages, detailed blockage covers a single shape. Therefore, detailed blockages inherit the color of the parent shape.



Related Topics

Color Representation for Pins

Supported Blockage Colors in Multi-Patterning Technology

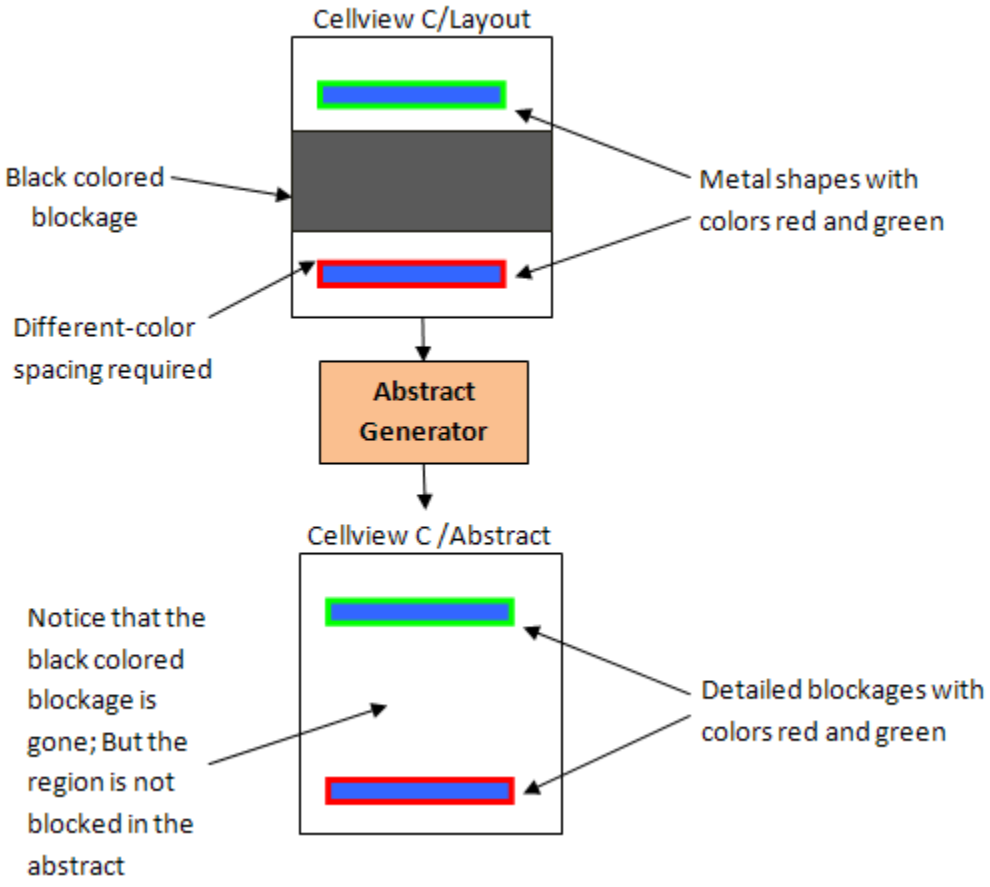
© 2023 Cadence Design Systems, Inc.

Supported Blockage Colors in Multi-Patterning Technology

Apart from the ‘multi’ and individual shape colors, Abstract Generator supports the following blockage colors.

- **Gray:** Pins and blockages corresponding to isolated shapes in the layout editor are assigned the gray color. Such pins can be colored, which means that their color is driven by their environment. Gray pins can be connected using any colored wire. During the Pins and Extract steps, if any gray metal shapes are coincident with any colored shape on the same layer, then the gray shape is dropped.
- **Black:** Blockages defined in the layout editor to keep routing out of a particular region are assigned the black color. Other colored shapes are kept at different-color spacing away from a black blockage.

During abstract generation, the region corresponding to a black blockage is kept clear of any shapes.



© 2023 Cadence Design Systems, Inc.

Related Topics

Color Representation for Pins

Color Representation for Blockages

© 2023 Cadence Design Systems, Inc.

Support for Trim Metal

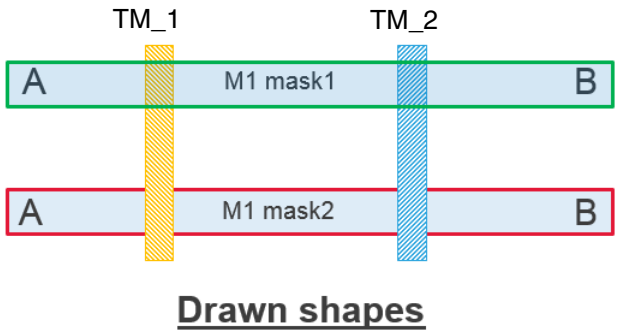
Trim metal, also referred to as cut metal, is a physical layer used to trim portions of a metal layer to achieve tight end-to-end line spacing. Trim metals are defined in the technology file. They can be assigned to cut-colored (masks) or non-colored metal geometries.

A trim metal trims a specific color (mask) of metal in the layout. For example, `TM_1` trims `Metal2:mask1` and `TM_2` trims `Metal2:mask2`. Therefore, in situations where a trim metal cuts multiple metal layers, only a specific metal layer is trimmed. Please note that metal layers are cut during fabrication.

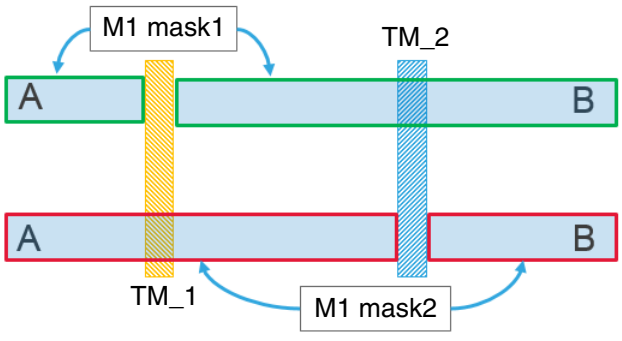
In the following diagrams, `TM_1` is a trim metal that cuts only `(Metal1 mask1)`; and `TM_2` cuts only `(Metal1 mask2)`. These trim metals:

- Do not have any effect on the same metal shapes drawn on other masks
- Do not affect any other metal

In the first figure, there are continuous metal lines in the layout. These shapes are cut into two parts by trim metals, which can be connected to different nets, as shown in the second figure.



Drawn shapes



Fabricated shapes

Trim metals can be assigned to:

© 2023 Cadence Design Systems, Inc.

- **Non–colored (gray) metal geometries:** You can specify the color of the metal layer to be trimmed.
- **Cut-Colored metal geometries:** Trims metal layers of the same color as the trim metal.

Abstract Generator supports trim metals. While loading a library, Abstract Generator automatically fetches information about trim metals from the technology file. Trim metals are recognized at each step of the abstract generation process. Therefore, at each step, metal layers are considered as effective after they are cut by trim metals in the previous step. Only metal shapes are cut by trim metals.

Abstract Generator supports coloring for all metal layer shapes up to `Mask8` and supports coloring for trim layers up to `mask3`.

The following table summarizes the effect of trim metals in each step of the abstract generation process:

Abstract Generation Step	Effect of Trim Metals
Pins	Pins under labels are cut.
Extract	Metal layers are cut during the following tasks: ■ Net extraction ■ Pin geometry restrictions ■ Antenna extraction
Abstract	■ Detailed blockages are cut. ■ Cover blockages are not cut.

Related Topics

Prerequisites for Multi-Patterning Technology Support

Color Representation for Pins

Color Representation for Blockages

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

A

Virtuoso Abstract Generator Forms

This topic lists the forms and windows that can be invoked through the integrated Abstract Generator and Standalone Abstract Generator.

- Abstract Generator Form
- Abstract Pcells Form
- Add User Bin Form
- Attach Technology Library Form
- Cell Properties Form
- Cell Report Window
- Copy Bin Options Form
- Delete User Bin Form
- Distribute Cells Among Bins Form
- Export LEF Form
- Export Options Form
- Export Report Form
- General Options Form
- G-Spec Builder Form
- Import DEF Form
- Import Layout Form
- Import LEF Form
- Import Logical Form
- Import Oasis Form
- Import Options Form
- Integrated Abstract Generator Form

© 2023 Cadence Design Systems, Inc.

- Mark/Unmark Pcells for Abstraction Form
- Move Selected Cells Form
- New Library Window
- Open Library Window
- Rename User Bin Form
- Replay Form
- Running step Abstract Form
- Running Step Extract Form
- Running Step Pins Form
- Save Dynamic Abstract Generation Options Form
- Select Cells Form
- Sort Cells Form
- Specify SKILL Files Form
- UnAbstract Pcells Form

© 2023 Cadence Design Systems, Inc.

Abstract Generator Form

Use the Abstract Generator form to generate an abstract of a Pcell in the layout window.

Field	Description
File	This menu includes the file related commands.
Tools	This menu includes the tool related commands.
Show Cell Report	Displays the status of the cells included in the abstract generation process.
Show Abstract Log	Displays log information of the current session.
Show Selected Cells	Displays a list of all the cells that are selected in the *Cell(s)* field in the Design Information section.
Design Information	This section includes various details that are specified in the design layout.
Library	Specifies the library containing the cells for which you want to create abstracts. The library list is read in from the `cds.lib` file located in the run directory, and libraries are listed in an alphabetical order.
Filter Cells	Lets you filter out cells based on their names. For example, type `top` to filter out cells that contain the string in their name.
Cell(s)	Lets you specify the cells for which you want to create abstracts. The *Cell(s)* list box displays the names of all the cells in the library selected in the *Library* drop-down list.
View	Lets you specify the view name in which you want to create abstracts.
Design Type	Indicates the type of cells to be processed in this abstract run: *Block* (default), *Core, IO*, and *Corner*.
Top Metal	Lets you specify the top metal layer. If you specify a *Top Metal* layer, abstract shapes are generated only for objects on the specified layer and its lower layers.
Options File	Lets you specify the path from where Abstract Generator loads the options.
Load Options	Loads the options from the specified options file.
Pins	This section extracts pin information.

© 2023 Cadence Design Systems, Inc.

Field	Description
Preserve Local Blockages	Controls how Abstract Generator processes local blockages present in the layout view.
Create Pins From Labels	Creates physical pin shapes corresponding to the geometry overlapping the text labels.
Label to Pin Map	Specifies the text labels. Abstract Generator creates pins by mapping overlapping geometries to text labels specified in this field. This field is enabled only when the *Create Pins From Labels* check box is selected.
Extract	Extracts connectivity data and calculates antenna information.
Signal	Extracts connectivity information for signal nets.
Power	Extracts connectivity information for power nets.
Calculate Antenna	Creates process antenna information for custom blocks and standard cells.
Connectivity	Specifies the connectivity between layers. This field is enabled only when one the three check boxes, *Signal*, *Power*, and *Calculate Antenna*, is selected.
Gate	Indicates the intersection of the first `Poly` layer with all diffusion layers in the antenna region. This field is enabled only when the *Calculate Antenna* check box is selected.
Drain	Indicates the drain identification layer in the antenna region. It is identified as the `andnot` of the first diffusion layer with all `Poly` layers. This field is enabled only when the *Calculate Antenna* check box is selected.
Signal Layers	Lists all signal layers to be extracted. This field is enabled only when the *Signal* or the *Calculate Antenna* check box is selected.
Pin Layers	Lists all the layers on which pins need to be created for signal nets. This field is enabled only when the *Signal* check box is selected.
Use Signal GSpec For Power GSpec	Extracts geometry specifications for the power layers. This field is enabled only when the *Power* check box is selected.
Power Layers	Lists all power layers to be extracted. This field is enabled only when the *Power* check box is selected.

© 2023 Cadence Design Systems, Inc.

Field	Description
Pin Layers	Lists all the layers on which pins need to be created for power nets. This field is enabled only when the *Power* check box is selected.
Abstract	Creates the final shapes required by place-and-route tools, and fractures pin shapes into rectangles.
Create Boundary Pins	Creates square pins at the boundary from the extracted pin geometries.
Signal	Creates boundary pins for signal nets.
Power	Creates boundary pins for power nets.
Create Overlap Boundary	Specifies whether a rectilinear PR boundary (overlap) should be generated during abstraction.
Signal Grouping	Controls how the pin geometries are grouped for corresponding signal nets. You can choose from *Single*, *Separate*, or *Overlap* (default).
Power Grouping	Controls how the pin geometries are grouped for corresponding power nets. You can choose from *Single*, *Separate*, or *Overlap* (default).
Cover Blockages	Specifies the layers to be considered while creating cover blockages.
Pin CutOut	Specifies the layers on which pin cutouts need to be created.
Fracture Blockages	Generates only rectangular geometries for blockages in the final abstract view.
Fracture Pins	Generates only rectangular geometries for pins in the final abstract view.
Generate Abstract	Generates abstracts for the selected cells.
Close	Closes the Abstract Generator form.

Related Topics

Generating Abstracts

© 2023 Cadence Design Systems, Inc.

Abstract Pcells Form

Use the Abstract Pcells form to generate an abstract of a Pcell in the layout window.

Field	Description
Selection	This section allows you to select the required Pcells in the layout canvas or from the Navigator assistant.
All Pcells	Creates abstracts for all the Pcells in the design.
Selected Pcells	Creates the abstracts for selected Pcells in the layout canvas. After you have run the command, you can select additional Pcells for abstraction, if required, and run the command again. A warning message is displayed stating that some Pcells could not be abstracted because the Pcells were unmarked for abstraction. A warning message is also displayed if abstract data already exists for a Pcell. To generate an abstract for such a Pcell, delete the abstract data by using the *Unabstract Pcells* command, and again abstract the required Pcells.
Options	This section lets you specify the settings for blockage generation and for loading or saving an options file.
Blockages	Lets you specify layers in the *Cover Blockage Layer*s field and specify a layer-geometry specification in the *Detailed Blockage Layers* field. When you type a value in a field in the *Blockages* group box, the checker running in the background validates the specified value. If the specified value is invalid, a pink highlight appears around the text box. A warning message is also displayed in CIW explaining why the value is invalid.

© 2023 Cadence Design Systems, Inc.

Field	Description
Cover Blockage Layers	Creates cover blockages for all layers and layer purpose pairs specified in the *Cover Blockage Layer*s field. For example if you specify `Metal1 Metal2 Metal3`, the cover blockage is created on all the three metal layers. By default, the *Cover Blockage Layers* field is empty. A warning message is displayed if the layers specified in the *Cover Blockage Layers* field are not complementary to the ones specified in the *Detailed Blockage Layers* field. For example, if you specify `Metal1` in the *Cover Blockages Layer* field and `Metal1 (Metal1 or Oxide)` in the *Detailed Blockage Layers* field, the following message is displayed: Cover blockage layers (`"Metal1"`) are already part of detailed blockage layer list (`"Metal1" "Oxide"`). Specify cover blockage layer list such that cover blockage layers are complementary to detailed blockage layers.
Detailed Blockage Layers	Creates detailed blockages for all layers and layer purpose pairs specified in the *Detailed Blockage Layers* field. The layer geometry specification that is displayed by default in the *Detailed Blockage Layers* field is determined by the technology file with which you are working. For example, if you specify `(Metal1 (Metal1 or Oxide)) (Metal2 (Metal2 or Oxide))`, the detailed blockage is created on `Metal1` and `Metal2` layer for, `Metal1` or `Oxide` and `Metal2` or `Oxide` layer-geometry in the design.
Load/Save Options File	Allows you to load or save the options file, which is an editable text file that contains the abstraction rules used by Dynamic Abstract Generation to create abstracts.
Options File Name	Specifies the name of the options file to which you want to save the abstraction rules in the *options File Name* field. You can also load an existing options file, edit it, and save the changes. By default, the name that appears is *.dynamicAbstract.options*.
Browse	Lets you browse the files and directly edit an existing options file in a text editor.
Load	Loads the various rules contained in an options file to the Abstract Pcells form. To load the abstraction rules contained in an options file, select the required options file and click *Load*.

© 2023 Cadence Design Systems, Inc.

Field	Description
Save	Saves the abstraction rules specified in the Abstract Pcells form to an options file.To save the abstraction rules to an options file, click *Save*.
Advanced Options	Lets you specify the advanced abstraction rules and opens the Save Dynamic Abstract Generation Options form.

Related Topics

Abstracting Pcells

Save Dynamic Abstract Generation Options Form

© 2023 Cadence Design Systems, Inc.

Add User Bin Form

Use the Add User Bin form to add a new user bin to the list of existing bins.

Field	Description
Enter bin name	Lets you specify the name of the new bin. The bin name should contain any spaces.
Choose bin type	Lets you select a bin from the drop-down list. The bin type determines the default options settings for the new bin. Therefore, for example, if you create a new user bin of type *Core*, the user bin options will be identical to those currently set for the default *Core* bin.

Related Topics

Adding a User Bin in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Attach Technology Library Form

Use the Attach Technology Library form to attach and associate existing technology library information to the current cell library.

Field	Description
Select Technology Library	Allows you to attach a technology library. This form lists all the libraries in the `cds.lib` file that have their own `techfile.cds` file contained in the library itself. The library list displayed also includes `cdsDefTechLib`, which is the default technology library. You cannot import a technology LEF file if the default technology file, `cdsDefTechLib`, is already attached to a cell library. If you attempt to do this, an error message will display.
Library	Lets you attach the library which Abstract Generator attempts to combine with the current library and loads it into the software. If you attempt to attach the same technology file that is already associated with the current library, you will be given the opportunity to reattach that technology library.
Path	Lets you specify the path of the library.

Related Topics

Attaching Technology Information to a Library in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Cell Properties Form

Use the Cell Properties form to you view and change the properties on cells.

Field	Description
Change property	Lets you select the property you want to change.
Apply	Updates the form and accepts all your changes.

Related Topics

Viewing and Editing Cell Properties

© 2023 Cadence Design Systems, Inc.

Cell Report Window

Use the Cell Report Window to view and change the properties on cells.

Field	Description
CellName	Displays a list of all the cells that are selected in the *Cell(s)* field in the *Design Information* section of the Abstract Generator form.
Status	Displays the status of the cells, green indicates no errors or warnings, orange indicates warnings, and red indicates errors during abstract generation.
Errors	Displays the number of errors that were reported during abstract generation.
Warnings	Displays the number of warnings that were reported during abstract generation.
Report	Displays the *View* button against every cell. Click the button to view detailed information about the abstract generation process, including the time taken for abstract generation, types of blockages created on different layers, and the complete list of errors and warnings that were reported.

Related Topics

Viewing the Cell Report

Viewing Results in the Abstract Generator Form

© 2023 Cadence Design Systems, Inc.

Copy Bin Options Form

Use the Copy Bin Options form to copy option settings from another bin to the selected bin. This is useful when creating a new user bin; you can selectively copy bin specific options only while copying options from one bin to another.

Field	Description
From	Lets you select the bin whose options you want to copy. The available options are *Core*, *IO*, *Corner*, and *Block*. For example, the settings for the *Core* bin could be copied to *UserBin*.
To	Lets you enter the current bin.

Related Topics

Setting Bin Options in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Delete User Bin Form

Use the Delete User Bin form to select the bin you do not need and want to delete. If the bin you are deleting is not empty, you will be prompted to specify where to put its cells before the bin is deleted.

Field	Description
Delete bin	Displays the name of the selected user bin.
Move its cell to	Lets you select the bin to which you want the cells of the deleted bin to be moved. The cells are moved and the user bin is deleted.

Related Topics

Deleting a User Bin in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Distribute Cells Among Bins Form

The Distribute Cells Among Bins form lets you distribute cells among the system and user bins according to cell height, name or type.

Tab	Description
Method	This section allows you to specify the criteria you want to use.
Height	Lets you specify a series of height ranges in μ, and the bins to which cells falling within a particular range are to be moved. To ensure that no cells are missed, only the lower bound of each range is editable. When you change the lower bound of one range, the upper bound of the next range down is adjusted automatically.
Name	Lets you specify strings of characters and target bins for cell names containing those strings. For example, the settings shown above would move all the cells with names that include the string *block* to the *Block* bin, those including the string *io* to the *IO* bin, and so on. Any cells not covered by specific criteria fall into the “other” category and are moved to the bin specified.
Type	Lets you distribute the cells to other system and user bins. The *Type* method distribution is based on the `CellType` attribute of a layout cell on OpenAccess. The first column represents the `CellType` attribute. By default, the `CellType` attributes in the first column are mapped to the corresponding system bins in the second column. The choices in the second column corresponding to a specific type are restricted to the user-defined and system bins of the same type. For example, if *myBin* is a user-defined bin of the type *Block*, *myBin* would be available for selection with the *Block* system bin in the second column. For cellviews with an unknown `CellType` (*Other*), all system and user-defined bins come up as options in the *Ignore* cyclic list.
Cells	This section allows you to specify whether you want to distribute all or selected cells when you open the form.
All	Distributes all the cells displayed in the *Cell* pane.
Selected	Distributes the set of selected cells displayed in the *Cell* pane.

© 2023 Cadence Design Systems, Inc.

Related Topics

Distributing Cells into Bins in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Export LEF Form

Use the Export LEF Form to generate LEF descriptions for cells in one or more bins.

Field	Description
LEF Filename	Specifies a path and name for the LEF file to export or use the *Browse* button to specify a location.

Export Geometry LEF Data	Lets you choose the geometry LEF to be exported. All the LEF information is written to the specified *LEF Filename*.
	
	The following example shows the variants of LEF data that can be exported from Abstract Generator:
	
	Geometry LEF data
	
	`PIN A`
	`  DIRECTION INPUT ;`
	`  PORT`
	`  LAYER METAL1 ;`
	`  RECT 0.715 1.845 1.105 2.490 ;`
	`  END`
	`END A`
	
	Tech LEF data
	
	`LAYER METAL1`
	`  TYPE ROUTING ;`
	`  DIRECTION HORIZONTAL ;`
	`  PITCH 0.56 ;`
	`  WIDTH 0.23 ;`
	`  SPACING 0.23 ;`
	`  RESISTANCE RPERSQ 0 ;`
	`  CAPACITANCE CPERSQDIST 0 ;`
	`  THICKNESS 0.1 ;`
	`  ANTENNALENGTHFACTOR 0.00208 ;`
	`  CURRENTDEN 0 ;`
	`END METAL1`

© 2023 Cadence Design Systems, Inc.

Field	Description

Export Tech LEF Data	Lets you choose the technology LEF to be exported. All the LEF information is written to the specified *LEF Filename*. The following example shows the variants of LEF data that can be exported from Abstract Generator:
	`LAYER METAL1`
	`  TYPE ROUTING ;`
	`  DIRECTION HORIZONTAL ;`
	`  PITCH 0.56 ;`
	`  WIDTH 0.23 ;`
	`  SPACING 0.23 ;`
	`  RESISTANCE RPERSQ 0 ;`
	`  CAPACITANCE CPERSQDIST 0 ;`
	`  THICKNESS 0.1 ;`
	`  ANTENNALENGTHFACTOR 0.00208 ;`
	`  CURRENTDEN 0 ;`
	`END METAL1`

Cell List Filename	Exports LEF information for only the specified cells. The list of cells, whose LEF information is to be exported, is specified in a cell list file, with one cell name per line. Specify the name of this cell list file in the field or use *Browse* to specify the file's location. This is an optional field. For related messages, see ABS-16103 and ABS-16104. If you specify a cell list file, the *Export LEF for Bin* option gets disabled.
LEF Version	Lets you select the version of LEF that you want to export.
Export LEF for Bin	Selects the bins for which you want to export the LEF data. You can export LEF for all the cells in your library or on a bin-by-bin basis.
Bus characters and *Divider character*	Lets you select the characters you want to use to represent buses (< > or [ ]) and pseudohierarchy (/ or |), respectively, in your LEF.

Related Topics

Exporting LEF in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Export Report Form

Choose *File – Export – Report* to display the Export Report Form. The report function outputs a summary of the cells in your library.

Field	Description
Report Filename	Lets you specify a path and name for the file to be exported.
Browse	Lets you specify the location for the file to be exported.

Related Topics

Exporting a Report in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Export Options Form

Use the Export Options form to save the export option settings for the current session.

Field	Description
Options Filename	Lets you specify a path and name for the file.
Browse	Lets you specify the location for the file to be exported.

Related Topics

Exporting Options File in Abstract Generator

© 2023 Cadence Design Systems, Inc.

General Options Form

Use the General Options form to view and change options that apply to the overall Abstract Generator. The form contains the following tabs.

Tab	Description
General	Provides the options to set the default bin, LEF version, and the purpose name(s) to be ignored during the flow steps.
Views	Specifies the file names that Abstract Generator would use while creating layout, logical, and abstract database files. The *Logical file name* field can also be used to specify an existing logical view (for example, `schematic` or `symbol`) from which logical information is to be picked up.

General

The following table describes the fields available on the *General* tab of the General Options form.

Field	Description
Constraint group	Specifies the name of the constraint group of the technology database. Abstract Generator works with the specified constraint group and derives the required information, such as the valid vias and valid routing layers, only from that constraint group. By default, the *Constraint group* field is empty. You can specify the constraint group name in the *Constraint group* field. If you have specified a constraint group name, then Abstract Generator will search for the rules in the specified constraint group. If the rules are not found in the specified constraint group, Abstract Generator displays an error. However, if constraint group name is not specified, Abstract Generator will search for the rules in the constraint group of type `LefDefaultRouteSpec`. If the constraint group of type `LefDefaultRouteSpec` is not found, then Abstract Generator will search for the rules in the constraint group named `LefDefaultRouteSpec`. If the rules are not found in either the constraint group of type `LefDefaultRouteSpec` or in the constraint group named `LefDefaultRouteSpec`, Abstract Generator displays an error.
Pin purpose name	Sets the purpose for pins on the final abstracts. The default purpose is `net`.

© 2023 Cadence Design Systems, Inc.

Field	Description
Default Bin	Specifies the default bin into which library cells are imported. This setting only applies to any *new* library cells that are being imported. Any cells that have been imported previously will remain in their original import location. Cells will only be placed in the *Default Bin* if they conform to the following criteria: ■ There is a valid layout view. ■ The cell is not a via. ■ The cell is not a site. If any of these criteria are not met, the cells will be placed in the *Ignore* bin.
LEF Version	Sets the version of LEF to be used by Abstract Generator. You can select from `5.4`, `5.5`, `5.6`, and `5.7`. The default version of LEF used by Abstract Generator is `5.7` for mature nodes and `5.8` for advanced nodes.

© 2023 Cadence Design Systems, Inc.

Field	Description
Ignored Purpose(s)	Allows you to specify the purpose name(s) that you want to be ignored when a layer name without a qualifying purpose name is specified for the following options in the Pin, Extract, and Abstract steps: ■ Pins step: ❑ *Map* tab: *Map text labels to pins* field ❑ *Boundary* tab: *Using geometry on layers* field ■ Extract step: ❑ *Signal, Power*, or *Antenna* tab: *Geometry Specification* column in the *Layer Assignment for Signal Extraction* table ■ Abstract step: ❑ *Adjust* tab: *Geometry Specification* column in the *Layer Assignment for Pin Geometry* table ❑ *Blockage* tab: *Geometry Specification* column in the *Layer Assignment for Blockages* table ❑ *Overlap* tab: *Using geometry on layers* field Abstract Generator allows you to include the shapes on the ignored purpose(s) during the Pin, Extract, and Abstract steps if you explicitly mention the layer and the purpose names in the geometry specification at the appropriate locations in the flow steps. By default, this option is prepopulated with the purpose name `boundary`, so that the `boundary` purpose is ignored by default in all the flow steps. Abstract Generator allows you to include the shapes on the ignored purposes during the Pin, Extract, and Abstract steps if you explicitly mention the layer and the purpose names in the option value in the various steps. For example, if you specify the purpose to be ignored as "`boundary`", a geometry specification containing the layerName "`M1`" would include all shapes on layer "`M1`" and exclude those on purpose "`boundary`". To include the shapes on the ignored purpose "`boundary`", you will need to change the geometry specification to "`M1 or M1 boundary`".
Show Bubble Help	Displays bubble help for various GUI options. It is selected by default. You can turn it off if required.

© 2023 Cadence Design Systems, Inc.

Field	Description
Copy Routing Constraints from layout	Enables Abstract Generator to copy the mixed signal routing constraints from the layout view to the abstract view.

Views

The following table describes the fields available on the *Views* tab of the General Options form.

Field	Description
Layout View Name	Specifies the view name for layout information. For example, if you import GDSII layout data into Abstract Generator, this is the name used for the database files that are generated.
Logical View Name	Specifies the view name for logical information. For example, if you import Verilog information into Abstract Generator, this is the name used for the database files that are generated. You can also use this field to specify the name of an existing view containing logical information (for example, schematic or symbol). Abstract Generator automatically picks up logical information from the specified file. If you do specify `schematic` or `symbol` in this field, you cannot then import Verilog data using the *Import Logical* form. The entry in this field must be compatible with the type of data that is subsequently imported.
Abstract View Name	Specifies the name used for the abstract views created by Abstract Generator.

Related Topics

Specifying Global Options in General Options form

© 2023 Cadence Design Systems, Inc.

G-Spec Builder Form

Use the G-Spec Builder form to build own geometry specifications (g-specs) using temporary layers. These temporary layers are then translated into the correct g-spec syntax.

Field	Description
Result	Specifies the layer on which the resulting shapes from the Boolean operation will be placed. You can do one of the following: ■ Select a layer that exists in the technology file. ■ Manually enter the name of a layer that exists in the technology file. ■ Manually enter the name of a temporary layer.
Layer 1	Specifies the layer of the layer-purpose pair being used for defining the geometry specification. You can do one of the following: ■ Select a layer that exists in the technology file. ■ Manually enter the name of a layer that exists in the technology file. ■ Manually enter the name of a temporary layer.
Purpose 1	Specifies an al field. As above, the layer or layer-purpose pair must be present in the technology file.
Operator	Selects an operator to use in the Boolean operation. The `size` operator accepts a float number rather than another layer-purpose pair as an argument. In this case, the layer field already defined in the technology file is editable.
Layer 2	Specifies the layer of the second layer-purpose pair being used for defining the geometry specification. You can do one of the following: ■ Select a layer that exists in the technology file. ■ Manually enter the name of a layer that exists in the technology file. ■ Manually enter the name of a temporary layer.

© 2023 Cadence Design Systems, Inc.

Field	Description
Purpose 2	Specifies an al field where you can specify the name of a purpose. The layer or the layer-purpose pair must be present in the technology file.

Related Topics

Specifying General Options in G-Spec Builder Form

© 2023 Cadence Design Systems, Inc.

Import DEF Form

Use the Import DEF form to import a net list in DEF format and update the design with the information.

Field	Description
DEF Filenames	Specifies the DEF files to import. You can type in the name manually or *Browse* for existing files with the `.def` extension in the current directory.
Default Bin	Lets you select the bin to which DEF data is to be imported. Cells are placed in the *Default Bin* only if they confirm to the following criteria: ■ There is a valid layout view. ■ The cell is not a via. ■ The cell is not a site. If any of this criteria is not met, the cells will be placed in the *Ignore* bin.

Related Topics

Importing DEF in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Import Layout Form

Use the Import Layout form to create cellviews from existing layout information in GDSII format.

Field	Description
GDSII Filenames	Lets you specify the names of the GDSII files that you want to import.
Map GDSII names	Lets you specify whether you want GDSII names mapped to upper case, lower case, or to not be mapped at all.
Layer Map Table	Lets you specify the name of the layer map file you want to import.
Default Bin	Lets you select the bin into which the cells are imported. This option applies only to any *new* library cells being imported. Cells imported previously are not moved. During import, Abstract Generator creates layout views for the cells in the library. Some types of cells, such as sites and generated vias, do not need to be processed, so Abstract Generator moves them directly into the *Ignore* bin. A cell is placed in the *Default Bin* only if it meets the criteria below. ■ There is a valid layout view. ■ The cell is not a via. ■ The cell is not a site

© 2023 Cadence Design Systems, Inc.

Field	Description
Template Filename	Lets you specify a stream template file to be imported into Abstract Generator. If you do not select a template file, Abstract Generator creates a default file called `PcsStreamIn.template` in the `.abstract/import` directory. This file has default option settings of *No Mapping* and *Core* bin and can be used to create your own template file. **Note:** Regardless of whether a template is supplied or not, the values of the *GDSII Filenames* and the *Map GDSII names* options are always the values specified in the form. See Chapter 2 of the Design Data Translator's Reference. The subsection `techControl`, added to the technology file for ITDB (Incremental Technology Database) support, necessitates the specification of a set of ordered reference libraries to allow inheritance from these libraries.
Technology Reference Library Name	Allows you to specify one or more valid reference technology libraries. The specified referenced technology libraries must be present in the `cds.lib` file and must be readable.

Related Topics

Importing GDSII Information in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Import LEF Form

Use the Import LEF form to create cellviews from LEF data.

Field	Description
LEF Filenames	Specifies the LEF files to import. You can type the name manually in the field or *Browse* for existing files with the `.lef` extension in the current directory.
View	Specifies the cellview to which a LEF macro will be saved. The default view is `abstract` but this can be changed so that the LEF macro is saved to another view, such as the `layout` view.
Technology Reference Library Name	Allows you to specify one or more valid reference technology libraries. The specified referenced technology libraries must be present in the `cds.lib` file and must be readable.
Overwrite	Enables Abstract Generator to overwrite the current technology file if the LEF file to be imported contains one. This option is off by default. All kinds of LEF files can be imported, ones with both technology and geometry information and also the ones with only technology or only geometry information. You cannot import a technology LEF file if the default technology file, `cdsDefTechLib`, is already attached to a cell library. If you attempt to do this, an error message will display.

© 2023 Cadence Design Systems, Inc.

Field	Description
Library Name	Lets you specify the name of the LEF library that needs to be imported. The library and its associated technology file are loaded into the current abstract session. At this point, the corresponding GDS file, if available, will also be read. When specifying the library name: ■ If a library is already loaded in the current abstract session, then the name of the currently open library is displayed in the *Library Name* field. Either retain the same library name or specify a different value. ■ If you have created a new library, then the name of the newly created library is displayed in the *Library Name* field. Either retain the same library name or specify a different value. ■ If no library is currently loaded in AG, and user wants to create new library using the Import LEF form, then it is mandatory to specify the library name along with the LEF file name. Else an error message is displayed and the Import LEF operation fails. ■ When you click *OK*, Abstract Generator opens the specified library. So, if you specify a library name other than the currently loaded library, then the newly created library is loaded. Alternatively use option `LEFLibraryName`.

Related Topics

Importing DEF in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Import Logical Form

Use the Import Logical form to create cellviews from existing logical information, typically represented in Liberty or Verilog® formats.

Field	Description
Logical Filenames	Specifies the name of one or more files you want to import, or use the *Browse* button to help locate the file and add it to the selection.
Filetype	Lets you filter the required file format by using a search list. Abstract Generator creates cells with logical views in the current library directory. If the cell already exists, Abstract Generator does not create a new cell but adds logical view information to the other views present for the cell. If you import logical information after you have run the Pins step, Abstract Generator confirms that the layout and logical views match before extracting terminal properties from the logical view. Cells that have only a logical view go to the *Ignore* bin by default.

Related Topics

Importing Logical Data in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Import Oasis Form

Use the Import Oasis form to create cellviews from existing layout information in OASIS format.

Field	Description
Oasis Filenames	Specifies the names of the OASIS files that you want to import. Use the *Browse* button to help you locate the files.
Map Oasis names	Lets you specify whether you want OASIS names mapped to uppercase, lowercase, or not to be mapped at all.
Layer Map Table	Specifies the name of the layer map file you want to import. Use the *Browse* button to help you locate the file.
Default Bin	Lets you select the bin into which the cells are imported. This option applies only to any *new* library cells being imported. Cells imported previously are not moved. During import, Abstract Generator creates layout views for the cells in the library. Some types of cells, such as sites and generated vias, do not need to be processed, so Abstract Generator moves them directly into the *Ignore* bin. A cell is placed in the *Default Bin* only if it meets the criteria below. ■ There is a valid layout view. ■ The cell is not a via. ■ The cell is not a site.
Template Filename	Specifies a stream template file to be imported into Abstract Generator. Use the *Browse* button to help you locate template files. If you do not select a template file, Abstract Generator creates a default file called `PcsStreamIn.template` in the `.abstract/import` directory. This file has default option settings of *No Mapping* and *Core* bin and can be used to create your own template file. Regardless of whether a template is supplied or not, the values of the *Oasis Filenames* and the *Map Oasis names* options are always the values specified in the form.

© 2023 Cadence Design Systems, Inc.

Field	Description
Technology Reference Library Name	Allows you to specify one or more valid reference technology libraries in the *Technology reference library name* field. The specified referenced technology libraries must be present in the `cds.lib` file and must be readable.

Related Topics

Importing OASIS Data in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Import Options Form

Use the Import Options form to load an options file saved in a previous session. You can save the settings of a session and load these settings back when you start a new session using the *Import* command.

Field	Description
Options Filename	Specifies the path and name of the file to import, or use *Browse* to look for the file through various folders.
OK	Imports the options file from the specified location.

Related Topics

Importing Options File in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Integrated Abstract Generator Form

Use the integrated Abstract Generator to generate abstracts while you are working on layout designs in the Virtuoso Studio design environment.

Field	Description
Pins	This is the first step of the abstract generation process.
Preserve Local Blockages	Controls how Abstract Generator processes local blockages present in the layout view. When unselected (default), Abstract Generator removes any local blockages, and proceeds as if they had never been specified in the DEF file. When selected, Abstract Generator preserves the local blockages, along with their attributes, from level 0 to level 32.
Create Pins From Labels	Creates physical pin shapes corresponding to the geometry overlapping the text labels. Additionally, specify the text labels in the *Label to Pin Map* field. Abstract Generator creates pins by mapping overlapping geometries to text labels specified in this field.
Extract	This is the second stage in the abstract generation process, involves extraction of new pins by tracing the physical connections starting from pins created in the *Pins* step. Antenna numbers are also calculated based on the options selected. You can use the progressive disclosure arrow to reveal or hide the options.
Signal, *Power*, *Calculate Antenna*	Choose whether you want to extract connectivity information for *Signal*, *Power*, or both types of nets.

Connectivity	Specifies the connectivity between layers. You can specify the layer connectivity without having to add via definitions in the technology file. The format for entering a value in this field is to list the layers as a number of entries; each entry being a combination of two or three layers.
	
	`(layer1 layer2 layer3) (layer4 layer5 layer6) ... (layer7 layer8) ...`
	
	Example:
	
	`(M1 M2 VIA1)(M2 M3 VIA2)(M4 M5)`

© 2023 Cadence Design Systems, Inc.

Field	Description
Calculate Antenna	Creates process antenna information for custom blocks and standard cells.
Gate	The *Gate* region is identified as the intersection of the first `Poly` layer with all diffusion layers. Example: (`Poly and Active`). Here, `Poly` is the gate identification layer.
Drain	The *Drain* region is identified as the `andnot` of the first diffusion layer with all `Poly` layers. Example: (`Active andnot Poly`). Here, the diffusion layer acts as the drain identification layer. The integrated Abstract Generator form supports only one Oxide model. This means that only the first diffusion layer is identified as the drain region. If there is more than one diffusion layer in the technology file, then the drain identification must be set through the options file. In this case, the form shows only the first drain region identification setting.
Signal Layers	Lists all signal layers to be extracted. Select *Use Signal GSpec for Power GSpec* to extract the geometry specifications for these layers.
Pin Layers	Lists all the layers on which pins need to be created for signal nets.
Power Layers	Lists all power layers to be extracted. Select *Use Signal GSpec for Power GSpec* to extract the geometry specifications for these layers.
Pin Layers	Lists all the layers on which pins need to be created for power nets. The *Signal Layers* and *Power Layers* list is controlled by the *Top Metal* layer specified in the *Design Information* section. For example, if *Top Metal* is set to `M3`, then only layer `M3` and its lower layers can be listed in the *Signal Layers* and *Power Layers* text boxes. Therefore, Abstract Generator can be run only on these layers.
Abstract	This section includes the options in the last step of the abstract generation process. You can use the progressive disclosure arrow to reveal or hide the options.

© 2023 Cadence Design Systems, Inc.

Field	Description
Create Boundary Pins	Creates square pins at the boundary from the extracted pin geometries. Choose whether you want to create boundary pins for *Signal* nets, *Power* nets, or both. Use this option to specify whether a rectilinear PR boundary (overlap) should be generated during abstraction. If the `.abstract.options` file is loaded, the layers specified in the `AbstractOverlapLayers` section are used to create the overlap boundary.
Create Overlap Boundary	Specifies whether a rectilinear PR boundary (overlap) should be generated during abstraction. If the `.abstract.options` file is loaded, the layers specified in the `AbstractOverlapLayers` section are used to create the overlap boundary.
Signal Grouping and *Power Grouping*	Controls how the pin geometries are grouped for corresponding signal and P/G nets. This is useful for blocks with numerous signal and/or power pins on different layers. *Signal Grouping* and *Power Grouping* are cyclic fields that provide the following options. ■ *Single*: Creates a single pin geometry for all net shapes. ■ *Separate*: Creates separate pin geometries for each net shape. ■ *Overlap*: Forms pin geometries from groups of overlapping net shapes. **Note:** Set a value for this option after you have selected the corresponding *Create Boundary Pins* option.
Cover Blockages	Specifies the layers to be considered while creating cover blockages. If a layer is removed from this list, then the layer is considered while created detailed blockages.
Pin CutOut	Lists the layers on which pin cutouts need to be created. Pin cutouts are applicable for all types of blockages: cover, detailed, and shrink. The list of layers is limited by the *Top Metal* specified in the *Design Information* section.
Fracture Blockages and *Fracture Pins*	Lets you generate only rectangular geometries for pins and blockages in the final abstract view. When the options are selected, all generated pins and blockages in the abstract view are fractured to generate only rectangular geometries. Otherwise, their shapes are retained.

© 2023 Cadence Design Systems, Inc.

Related Topics

© 2023 Cadence Design Systems, Inc.

Mark/Unmark Pcells for Abstraction Form

Use the Mark/Unmark Pcells for Abstraction form form to mark or unmark Pcells for abstraction.

Field	Description
Mark	Marks the Pcells for abstraction.This is also the default option.
Unmark	Unmarks a Pcell for abstraction. As a result, the Pcell is ignored for the abstraction process.
All Pcells	Abstracts all Pcells in the layout window.
Selected Pcells	Abstracts the selected Pcells in the layout window.

Related Topics

Marking or Unmarking Pcells for Abstraction

© 2023 Cadence Design Systems, Inc.

Move Selected Cells Form

Use the Move Selected Cells form to moves one or more selected cells to a different bin.

Field	Description
To bin	Lets you choose the target bin and move the selected cells into the selected bin. The status for all the views remains unchanged.

Related Topics

Moving Cells Across Bins in Abstract Generator

© 2023 Cadence Design Systems, Inc.

New Library Window

Use the New Library window to create a new library in the specified directory. You can create a library at any time during the abstract generation process.

Field	Description
Library Path	Specifies a valid directory path to where the new library should be created. The default value is the current directory, that is, ".".
Browse	Displays the Browse New Library Directory form that can be used to locate the required directory location for the new library. If no path is specified, a warning message is displayed requesting that you enter a valid directory path.
Library Name	Specifies a valid library name. The library name must not contain any spaces. You also cannot specify the name of an existing library that is already specified in the `cds.lib` file of the current directory.

Related Topics

Creating a Library in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Open Library Window

Use the Open Library window to specify the library that you want to process in Abstract Generator.

Field	Description
Select Library	Selects the library that you want to open. All the views created by Abstract Generator, including the final abstracts, are stored in this library.

Related Topics

Opening a Library in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Rename User Bin Form

Use the Rename User Bin form to change the name of the selected user bin.

Field	Description
Enter new bin name	Lets you specify the new name for the selected user bin.

Related Topics

Renaming a User Bin in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Replay Form

Use the Replay form to replay any file in either GUI or non-GUI mode.

Field	Description
Replay Files	Specifies the name of the file to be replayed.
Browse	Helps you to locate the file.

Related Topics

Replaying Commands in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Running Step Pins Form

Use the Running step Pins form to derive pins from the text labels in the layout view and place the locations at the text origins.

This form contains the following tabs.

Tab	Description
Map Tab	Specifies the text labels to be used for creating starting pin shapes for each of the nets to be extracted. You can also specify the pins that are power, ground, clock, analog, or output pins.
Text Tab	Allows you to change the name of the pins created in the Pins view from the original value in the layout. For example, the expression "`\\.extra.*`" would create a pin of VDD from a text label of VDD.extra1. by specifying the appropriate rule on the *Text* tab.
Boundary Tab	Controls the calculation of the place-and-route (PR) boundary, which defines the size of a single cell or block. In addition to specifying whether or not the PR boundary is to be created, you can specify the layers to be used for calculating the PR boundary and the details for adjusting or fixing the size of the calculated boundary.
Blocks Tab	Allows you to use the options on the tab with any layout view that has existing routing. Typically, these will be routed blocks imported from DEF.

© 2023 Cadence Design Systems, Inc.

Map Tab

The following table describes the fields available on the *Map* tab of the Running step Pins form:

Field	Description

Map text labels to pins	Specifies the text labels in the layout view that are to be mapped to the pins on nets in the pins view. This field is case-sensitive.
	
	The syntax is as follows:
	
	`(textLPP1 geomLPP1 geomLPP2)`
	`(textLPP2 geomLPP3 geomLPP4)`
	
	where,
	
	■ The syntax of `textLPP` is either `(textLayer textPurpose)` or `textLayer`.
	
	■ The syntax of `geomLPP` is either `(geomLayer geomPurpose)` or `geomLayer`.
	
	If you specify only a layer name instead of a layer-purpose pair, Abstract Generator considers all the purposes on that layer excluding the ones specified in the *Ignored Purpose(s)* field on the *General* tab (*File – General Options*).

Power pin names, Ground pin names, Clock pin names, *Analog pin names*	Specifies the pin names in the *Power pin names*, *Ground pin names*, *Clock pin names*, *Analog pin names*, and *Output pin names* fields. Abstract Generator identifies the respective pin names based on the label strings in the abstract.

© 2023 Cadence Design Systems, Inc.

Field	Description
Exclude existing terminals	Specifies a list of existing terminals that you want Abstract Generator to ignore. You can specify the terminal names in the form of regular expressions and separate the terminal names in the list by using blank spaces. To ignore all the terminal names, type `.*`. By default, this field is empty. Abstract Generator removes the specified terminals. In addition, Abstract Generator creates the deleted terminals afresh from the equivalent text labels existing in the layout. Abstract Generator issues the warning message ABS-12002 if a deleted terminal is not created from a text label existing in the layout. The new terminal created by Abstract Generator does not inherit any attributes or properties from the equivalent deleted terminal. Alternatively, you can use the `ExcludeExistingTerminals` option to ignore the terminals. `absSetBinOption("Core" "ExcludeExistingTerminals" "Y")` Replace `Y` with the terminal names you want to ignore.
Exclude existing pins on layers	Specifies a list of layers on which you want Abstract Generator to ignore the existing pins. You can specify the layer names as a list by separating the individual layer names using blank spaces. Abstract Generator will remove all the pins on the specified layers. If Abstract Generator comes across terminals that do not have any pins, it will issue the warning message ABS-12027 and remove those terminals. Next, Abstract Generator will recreate the deleted pins and any of the deleted terminals afresh from the equivalent text labels existing in the layout. The new pins (and terminals) created by Abstract Generator will not inherit any attributes or properties from the equivalent deleted pins. By default, the *Exclude existing pins on layers* field is blank.
Specify the top metal layer for cover blockage	Lets you create cover blockages. The cover blockages that are annotated to the layout view are in the form of properties or extensions, and they are not visible as normal blockages in the layout view. See Layout Dual View Support.

© 2023 Cadence Design Systems, Inc.

Text Tab

The following table describes the fields available on the *Text* tab of the Running step Pins form.

Field	Description
Label search depth	Controls how far down the design hierarchy does Abstract Generator search for text labels during text-to-pin mapping. The default value is `0`. This means that Abstract Generator only looks at text labels in the top level of the hierarchy and so only those labels could be used to create starting pin shapes for the extractor. Sometimes, the layout can be created with labels deeper in the hierarchy. If this is the case, you can change this setting accordingly. It is, however, recommended that you do not change this value unless absolutely necessary. There are often many labels at lower hierarchy levels that are *not* intended to become pins in the top level of the abstract.
Geometry search depth	Controls how far down the design hierarchy does Abstract Generator search for metal underneath a text label. The default value is `20`. However, if you know that the metal shapes that you want to map to are located at the top of the hierarchy, you can set the option appropriately. This should speed up processing time in the Pins step because only top level shapes will be stored in the geometry cache.
Text Manipulation	This section ensures that the pin names created during the Pins step match the logical pin names. Text labels often include special text at the end of the label that is not required for pin naming. The function uses regular expressions to locate unnecessary text in labels and either removes it completely or replaces it with the text you require.

© 2023 Cadence Design Systems, Inc.

Field	Description
Remove Text	Lets you specify a regular expression representing the pin label text that you are searching. ■ `\\.extra.*` Specifies that "`.extra.`" followed by any other character(s) (represented by `*`) is to be removed. For example, `VDD.extra1`, `VDD.extra2... VDD.extra20`, and so on are all converted to `VDD` if the *Replace With* field is left blank. ■ `:.` Specifies that "`:.`" followed by any other character(s) is to be removed. For example, `VDD:P` and `VDD:Fred` are both converted to `VDD` if the *Replace With* field is left blank. ■ `\(([0-9]+)\)and \[([0-9]+)\]` Specifies that the characters `(` and `[` are to be converted to < and the characters `)` and `]` are to be converted to >. The *Replace With* field displays `<\1>` in both cases, where `\1` indicates that the *Replace With* text is the first expression matched in parentheses.
Replace With	Contains the replacement text for the *Remove Text* option. The replacement regular expression text is applied to all matching instances.
Add	Adds a new row to the table.
Remove	Deletes the selected rows from the table.
Preserve text labels	Retains all existing pins in the layout, along with their labels, irrespective of whether the pins were created by Abstract Generator during the Abstract step. This option is selected by default. Alternatively, you can use the `PinsTextPreserveLabels` option to preserve pins and their labels. `absSetBinOption("Core" "PinsTextPreserveLabels""true")` Ensure that the layer of the text label is included in `PinsText-PinMap`. See Pin Annotation.

© 2023 Cadence Design Systems, Inc.

Field	Description
Restrict Pins to PR Boundary	Controls the geometry of N-Well pins created during the abstract generation process. If selected, the size of N-Well pins is restricted to the PR boundary. See N-Well and P-Well Pins Creation.

Boundary Tab

The following table describes the fields available on the *Boundary* tab of the Running Step Pins form.

Field	Description
Create boundary	Lets You create a PR boundary by choosing one of the values from *always*, *as needed*, and *off*. The new boundary is created as an OA `prBoundary` object. When checking for the existence of a PR boundary, Abstract Generator looks for a PR boundary object. If the PR boundary object is not found, then Abstract Generator searches for the shapes on layer `prBoundary` and purpose `boundary`. ■ *always*: Specifies that Abstract Generator creates a PR boundary after deleting any already existing PR boundary. Abstract Generator first looks for PR boundary object, deletes it, and then creates the new PR boundary object. ■ *as needed*: Specifies that Abstract Generator first checks whether a PR boundary exists. It creates a PR boundary only if no PR boundary already exists, for which it searches for PR boundary object in layout. Abstract Generator creates a rectangular PR boundary in the case of both *always* and *as needed* options. To obtain a rectilinear PR boundary, you can use the `PinsCreatePolyPRB` option. See Abstract Generator SKILL Functions. ■ *off*: Specifies that Abstract Generator never creates a PR boundary, though it checks whether or not a PR boundary exists and reports an error if there is no PR boundary in the layout unless the *Fix Boundary To* options are specified.

© 2023 Cadence Design Systems, Inc.

Field	Description
Using geometry on layers	Lets you specify the layers to be used to calculate the PR boundary. The boundary is drawn so that it encloses all the geometry found on these layers. For example, `metal1 (metal2 drawing)` This specifies all geometry on `metal1` all purposes except the ignored purposes and on `metal2` purpose `drawing` to be considered for PR boundary calculation.
Adjust Boundary By	Lets you increase or decrease the size of an existing or calculated PR boundary on one or more sides. To do this, specify the distance to be added to each boundary edge. For example, if you specify a positive number for the *Left* edge, the left boundary will be moved further to the left by that amount, that is, the lower left coordinate of the boundary will be decreased.
Fix Boundary To	Lets you fix individual cell boundary edges to an absolute value. These values are applied to existing and calculated PR boundaries. For example, you have two cells in a bin one of which already has a boundary and the other does not. If you set the *Create boundary* option to *as needed* and specify `-15` in the *Bottom* field of the *Fix Boundary To* section, Abstract Generator will automatically calculate the boundary for the other cell and set the lower edge of both cells to `-15` µ.

© 2023 Cadence Design Systems, Inc.

Blocks Tab

The following table describes the fields available on the *Blocks* tab of the Running Step Pins form.

Field	Description
Preserve local blockages in routed blocks	Controls how Abstract Generator processes local blockages defined in the `BLOCKAGE` section in DEF 5.8. Local blockages do not have the `+PUSHDOWN`, `+FILL`, or `+SLOT` modifiers. ■ When unchecked, Abstract Generator removes any local blockages and proceeds as if they had never been specified in the DEF file. ■ When checked, Abstract Generator preserves the local blockages along with their attributes for all the layers. The blockages along with their attributes are preserved from level 0 to level 32. The preserved layout blockages exists in the abstract view along with the other blockages generated in the abstract step. The layout blockages are not merged with the blockages generated in the abstract step and are separately maintained. This is done to enhance the performance.
Create power pins from routing	Controls whether pins are created from the routing defined in the `SPECIALNETS` section of the DEF file. When unchecked, Abstract Generator creates pins only from the pin geometry defined in the `PINS` section of the DEF file. If there is no pin geometry, the routing is turned into pins instead.
Using geometry on layers	Creates pins from the routing defined in the `SPECIALNETS` section for the layers specified in the *Using geometry on layers* field when *Create power pins from routing* is checked. It does this even if there are pins already defined in the `PINS` section. When creating pins, Abstract Generator ignores `FILLWIRE` and `SHIELD` net segments. It also creates pins in instances that are logically connected to the top-level nets.

© 2023 Cadence Design Systems, Inc.

Related Topics

Specifying Pin Mappings for Abstract Generation

Customizing Text Labels for Abstract Generation

Setting PR Boundary Creation Options in Standalone Abstract Generation

Customizing Abstracts of Routed Blocks in Standalone Abstract Generator

© 2023 Cadence Design Systems, Inc.

Running step Abstract Form

The Running step Abstract form allows you to modify only the options that are relevant to the abstract steps about to be run.

The form contains the following tabs.

Tab	Description
Adjust	Controls how pin shapes are adjusted during connectivity extraction for creating the final required pin shapes. These pin shapes are then fractured into rectangles. The options help you ensure that if a pin shape is reduced in any way, the discarded part of the pin is turned into a blockage, and the blockage is spaced sufficiently away from the reduced pin shape.
Blockage	Controls the final blockage geometry in the abstract view. Any blockage geometry already existing in the view is preserved and added to the final calculated blockages.
Density	Provides the specifications to Abstract Generator for calculating metal density (in percentage) for macros.
Fracture	Controls whether or not pins and blockages are fractured depending on the PR system you are using. By using the options on this tab, you can also control the modeling of 45-degree geometry. These options are available to all bins.
Site	Notifies the placer where it can place cells. All cells located in a particular bin share a common set of properties: symmetry, class (for example, pad or core), width, and height. These properties are used by the placer to assist in the placement process.
Overlap	Controlswhether Abstract Generator is able to update the overlap boundary geometry.

© 2023 Cadence Design Systems, Inc.

Adjust

The following table describes the fields available on the *Adjust* tab of the Running Step Abstract form.

Field	Description
Signal Nets	This section provides options to control how pin shapes are adjusted during connectivity extraction to create the final required pin shapes for signal nets.
Create boundary pins	Lets you adjust the pin shapes created during extraction to be square and on the cell boundary. This can be useful when processing blocks where pin geometry extends deep inside the block after extraction. This option exists at two locations on the *Adjust* tab — *Signal Nets* and *Power Nets* groups. By default, this option is off for all bins except the *Block* bin and is disabled and off for *Core* bin. If this option is turned off, the pin will retain the shape generated by the Extract step, which can still be modified using the *Layer Assignment for Signal Extraction* table options. The method used by Innovus to check width-dependent spacing rules might lead to false violations, so you should restrict the pin geometry to be a square near the boundary of the block by using the *Create boundary pins* option.
Boundary pin max distance to boundary	Allows you to specify the maximum distance (in microns) of the signal net or power net geometries from the boundary so that they can be considered as boundary pins. The geometries that are within the specified distance from the boundary are squared. By default, the shape(s) closest to the boundary is squared. This option is available only when the *Create boundary pins* option is selected. This option is present in both the *Signal Nets* and *Power Nets* groups on the *Adjust* tab

© 2023 Cadence Design Systems, Inc.

Field	Description
Signal geometry groups	Controls how signal net shapes are grouped into LEF PORTS. This is useful for blocks with numerous signal pins on different layers. Usually, the presence of overlapping shapes on different layers means that the router should connect to only one of the overlapping shapes. This is achieved by grouping the overlapping shapes into the same LEF PORT. This field provides the following options: ■ *single:* Creates a single LEF `PORT` for all the signal net shapes ■ *separate:* Creates a separate LEF `PORT` for each signal net shape ■ *overlap:* Forms LEF `PORTS` from groups of overlapping signal net shapes, that is, signal net shapes that overlap go into the same LEF `PORT`. The default value for this option is *single* for *Core*, *IO*, and *Corner* bins and *overlap* for *Block* bins. **Note:** Set a value for this option after you have selected the *Create boundary pins* option for signal nets.
Power Nets	This section provides options to control how pin shapes are adjusted during connectivity extraction to create the final required pin shapes for power nets.
Create boundary pins	Lets you adjust the pin shapes created during extraction to be square and on the cell boundary. This can be useful when processing blocks where pin geometry extends deep inside the block after extraction.
Boundary pin max distance to boundary	Allows you to specify the maximum distance (in microns) of the signal net or power net geometries from the boundary so that they can be considered as boundary pins. The geometries that are within the specified distance from the boundary are squared. By default, the shape(s) closest to the boundary is squared.

© 2023 Cadence Design Systems, Inc.

Field	Description
Create ring pins	Creates ring pins where any extreme shapes, that are close to the cell boundary existing in a power net are found to form a ring. This option is available only if you are processing a block cell. The selection of this option is also mutually exclusive from the *Create boundary pins* option. By default, this option is turned off. To create ring pins for these power rings, you must ensure that this option, along with the *Extract power nets* option on the *Power* tab in the Extract step, is turned on. If a block cell is processed with this option selected, Abstract Generator displays a message box informing you whether or not the ring pins were successfully created.
Ring pin max distance to boundary	Specifies (in microns) the maximum distance within which the extracted power net geometry can be from the boundary in order for it to be considered as part of the ring, and therefore, be included in the abstract as pin geometry. This field is only available if you are processing a block cell and when the *Create ring pins* option is already selected. Any `vdd` and `vss` net geometry found further than the specified distance from the boundary is not considered part of the ring geometry but is instead treated as an obstruction. The default value for this field is `0`, which means that Innovus finds the closest power shape to the boundary for each net, and that becomes the criterion distance used in determining the ring pin geometry for that net.

© 2023 Cadence Design Systems, Inc.

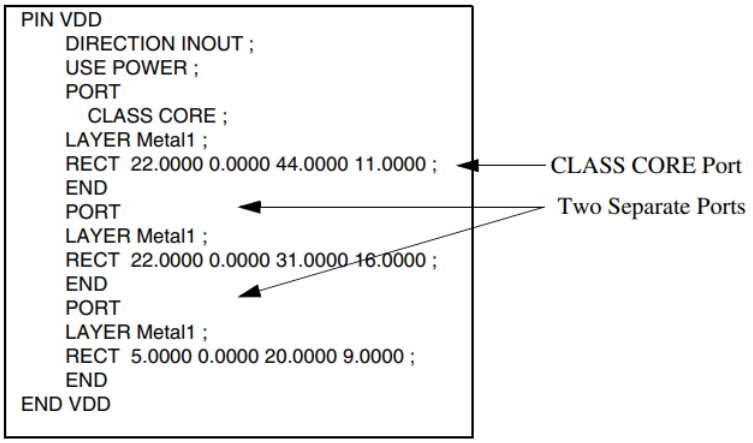
Field	Description
Follow ring pin	Considers all pin shapes, including the ones that are not part of the ring, as pins in the final abstract. This option is available only if you are processing a block cell and when the *Create ring pins* option is already selected. The pins that are not part of the ring (‘follow rings pins’) are included in the final abstract based on the value specified in the *Ring pin max distance to boundary* field. When no value is specified, only the pin shapes whose distance from the boundary is the same or less than the distance of the farthest edge of the ring pin shape are included in the abstract. On the other hand, when a value is specified in the *Ring pin max distance to boundary* field, pin shapes are included in the abstract only if their distance from the boundary is within the distance specified in the field. To prevent any filtering based on the distance, you can specify an arbitrarily large value in the *Ring pin max distance to boundary* field.

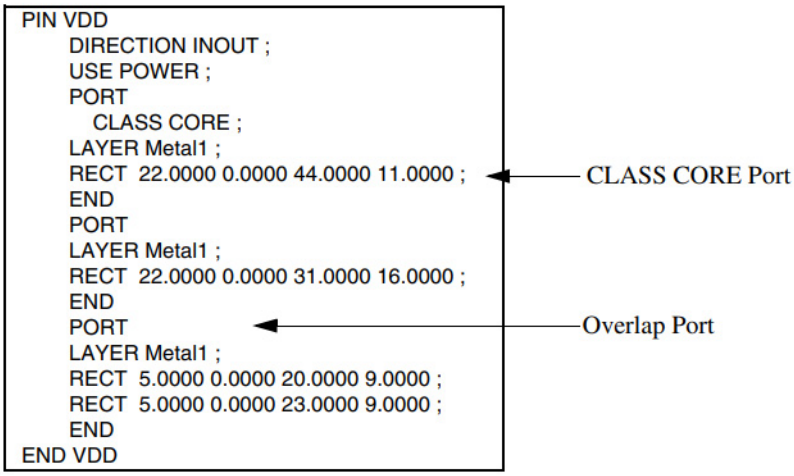
© 2023 Cadence Design Systems, Inc.

Field	Description
Power geometry groups	Controls how power shapes are grouped into LEF `PORTS`. This is useful for blocks with numerous power pins on different layers. Usually, the presence of overlapping power shapes on different layers means that the router should connect to only one of the overlapping shapes. This is achieved by grouping the overlapping shapes into the same LEF `PORT`. *Power geometry group* is a cyclic field and provides the following options: ■ *single:* Creates a single LEF `PORT` for all the power shapes ■ *separate:* Creates a separate LEF `PORT` for each power shape ■ *overlap:* Forms LEF `PORTS` from groups of overlapping power pins, that is, power shapes that overlap go into the same LEF `PORT`. The default value for this option is *single* for *Core*, *IO*, and *Corner* bins and *overlap* for *Block* bins. **Note:** Set a value for this option after you have selected the *Create boundary pins* option for power nets. **Examples** PIN VDD DIRECTION INOUT ; USE POWER ; PORT CLASS CORE ; LAYER Metal1 ; RECT 22.0000 0.0000 44.0000 11.0000 ; ← CLASS CORE Port END PORT ← Single Port LAYER Metal1 ; RECT 22.0000 0.0000 31.0000 16.0000 ; RECT 5.0000 0.0000 20.0000 9.0000 ; END END VDD

© 2023 Cadence Design Systems, Inc.

Field	Description

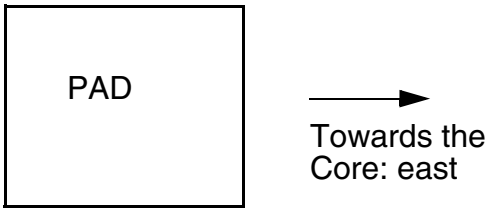




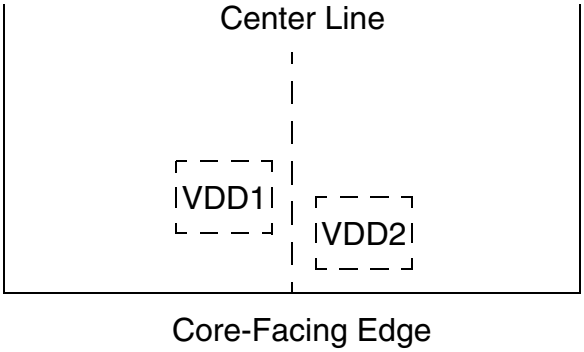
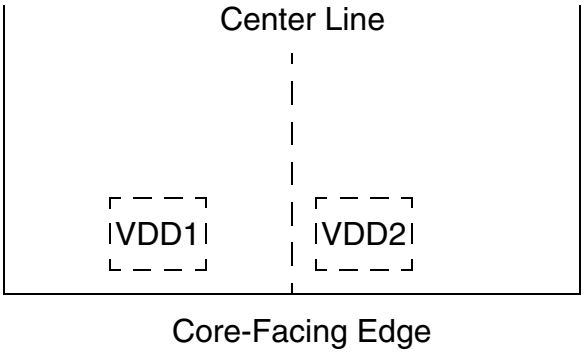
© 2023 Cadence Design Systems, Inc.

Field	Description
Power rail widths, offsets and shape	Provides an option to specify the characteristics of the power and ground rails for the current bin. The Abstract step does not assign feed through or abutment shape properties if there are no power terminal shapes that touch the PR boundary. For example, a voltage translator cell could have `vdd` and `vss` net geometry, but may also have an internal `vdd` net that cannot be classified under any shape class. You can set the shape manually using Viewing and Editing Cell Terminal Properties in Abstract Generator. Abstract Generator also examines the standard cell power pin shapes and sets the shape to either feed through or abutment. Additionally, checks are made at this time to ensure the following: ■ The selected standard cells have the same power rail width and the same ground rail width. ■ The selected standard cells have the power rails at the same offset from the cell origin so that the power rails can be connected by feed-through or abutment. ■ The rail width is the same at the right and left edges of standard cells. ■ The rail width is an even number when converted to LEF database units.
CORE/BUMP Ports	This section provides you options to specify the following CLASS CORE-specific information to Abstract Generator. These options apply only to the cells in the *IO* bin.

© 2023 Cadence Design Systems, Inc.

Field	Description
Cell edge facing core	Specifies the direction in which the `CORE`-facing edge is pointing, *north*, *south*, *east*, *west,* or a combination of any of the four core edges. The default value is *north*. You can specify multiple `CORE`-facing edges. The is a text field, which is enabled only when you process a pad cell in the *IO* bin. Abstract Generator uses this value to determine the edge of the pad cell that it should check for locating the port to be classified as `CLASS CORE` in the LEF file. For example, the IO pad shown in the following figure will be placed to the left of the core. Therefore, the *Cell edge facing core* option for this pad should be set to *east* so that the correct port is identified as CLASS CORE in the LEF file. 
Power/ground net to have CLASS CORE ports	Allows you to specify the nets that should have `CLASS CORE` ports. This can be specified by entering a regular expression in the field. This field is related with the *Cell edge facing core* option and therefore, available only when you process a pad cell in the *IO* bin.. The default entry for this field is `^((V(DD|CC))|(v(dd|cc)))(!)?$`

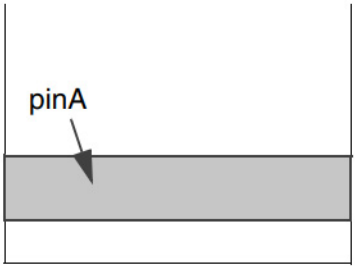
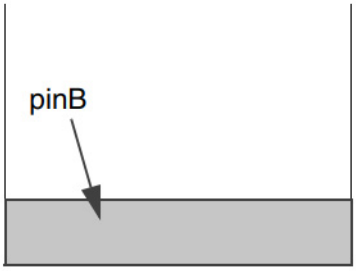
© 2023 Cadence Design Systems, Inc.

Field	Description
Allow multiple CLASS CORE ports	Specifies that two ports equidistant from the core-facing edge of a pad are both assigned the `CLASS CORE` statement in LEF. This option is intended for pad cells processed in the *IO* bin. Only geometries with center lines perpendicular to the core-facing edge are considered. The option is off by default, which means that only the geometry closest to the core-facing edge becomes a `CLASS CORE` port. In the figure below, the geometry that becomes a `CLASS CORE` port is the one on the right (`VDD2`).  When the option is off and there are two power geometries equidistant from the core-facing edge, only the geometry closer to the center line of the pad is defined as a `CLASS CORE` port in LEF. In the figure below, the geometry that becomes a `CLASS CORE` port is the one on the right (`VDD2`). 

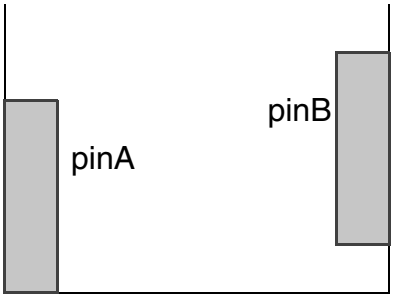
© 2023 Cadence Design Systems, Inc.

Field	Description
	When you select this option, the behavior depends on the Power geometry groups setting, as described below: ■ If you select *separate*, the geometries shown in the second picture above appear in two separate `CLASS CORE` ports in LEF. ■ If you select *single*, they appear in the same `CLASS CORE` port. ■ If you select *overlap*, they appear in two separate `CLASS CORE` ports *unless* they overlap, in which case they will appear in the same `CLASS CORE` port.
Copy CLASS CORE ports	Duplicates any geometry found to belong to a `CLASS CORE` port on the net (set by using the *Power/ground net to have CLASS CORE ports* option) in a non-`CLASS CORE` port to the exported abstract LEF. This option is enabled only when you process a pad cell in the *IO* bin and is switched off by default. When this option is off, Abstract Generator will find the power geometry group port (*single*, *separate*, *overlap*) whose geometry is closest to the core-facing edge and set it to `CLASS CORE`. Then, when exported to LEF, the port will only be represented once with the `CLASS CORE` property.

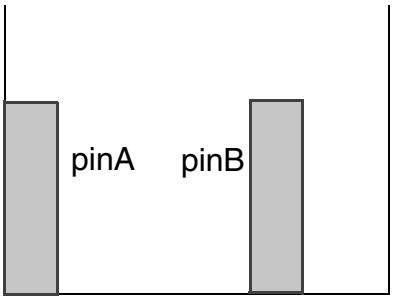
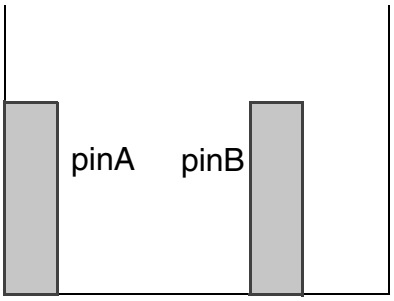
© 2023 Cadence Design Systems, Inc.

Field	Description
Create CLASS CORE ports only if pin meets cell boundary	Specifies that only the pins that either touch or cross the core-facing edge of the boundary become `CLASS CORE` ports in LEF. If the option is not selected, the pin closest to the core-facing edge becomes the `CLASS CORE` port. Consider the situation as shown in the figure below. In this situation, if the option is selected, then only `pinB` will become a `CLASS CORE` pin.  Core-Facing Edge  Core-Facing Edge Consider a situation in which the cell-edge facing core is set to south so the VDD pin on the right of the south edge becomes CLASS CORE in the LEF. Here is how this would look in the LEF file: PIN VDD DIRECTION INOUT ; USE power ; PORT CLASS CORE ; LAYER Metal1 ; RECT 40.280 0.000 42.120 17.680 ; END PORT LAYER Metal2 ; RECT 0.400 9.600 43.600 19.600 ; RECT 0.400 25.200 43.600 35.600 ; RECT 0.400 47.200 43.600 49.600 ; RECT 0.400 56.400 43.600 58.800 ; LAYER Metal1 ; RECT 1.880 0.000 3.720 10.480 ; END END VDD

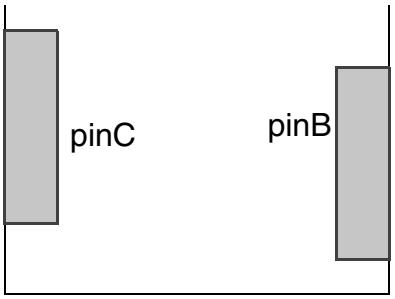
© 2023 Cadence Design Systems, Inc.

Field	Description
Create CLASS CORE ports if pin meets non-core facing edge	Allows the pins that touch the non-core facing edge — left or right (when the core-facing edge is north or south) to become eligible to be included as CLASS CORE ports. This option is switched on by default. The eligible pins become CLASS CORE ports if they satisfy other criteria (settings of other options on the *Adjust* tab). When this option is off, pins touching the non-core facing edge are not considered for the CLASS CORE port creation. ■ *Create CLASS CORE ports only if pin meets cell boundary* = ON  Assume that both pinA and pinB touch non-core facing edges. With the option *Create CLASS CORE ports if pin meets non-core facing edge* selected, both pinA and pinB are eligible to be considered as CLASS CORE ports. Since in this situation the *Create CLASS CORE ports only if pin meets cell boundary* option is also ON, only the pin touching the non-core facing edge that also touches the core-facing edge becomes the CLASS CORE port, which in this case is pinA. In the above figure, only pinA touches the non-core facing edge. However, since both *Create CLASS CORE ports if pin meets non-core facing edge* and *Create CLASS CORE ports only if pin meets cell boundary* options are ON, both pinA and pinB become the CLASS CORE ports.

© 2023 Cadence Design Systems, Inc.

Field	Description
	■ *Create CLASS CORE ports if pin meets non-core facing edge* = ON *Create CLASS CORE ports only if pin meets cell boundary* = ON  Core-Facing Edge ■ *Create CLASS CORE ports if pin meets non-core facing edge* = OFF *Create CLASS CORE ports only if pin meets cell boundary* = ON  Core-Facing Edge In the above figure, though pinA touches the non-core facing edge but it will not be considered for creating CLASS CORE port because the *Create CLASS CORE ports if pin meets non-core facing edge* option is OFF. In this case, with the *Create CLASS CORE ports only if pin meets cell boundary* option ON, only pinB becomes the CLASS CORE port.

© 2023 Cadence Design Systems, Inc.

Field	Description
	■ *Create CLASS CORE ports if pin meets non-core facing edge* = ON *Create CLASS CORE ports only if pin meets cell boundary* = OFF In this situation, only the pin touching the non-core facing edge that is also the closest to the core-facing edge becomes the CLASS CORE port. In the following figure, the pin that qualifies this criteria is pin B. 
Create CLASS BUMP ports	Considers power nets of cell type IO for creating the CLASS BUMP ports. The largest shapes (with the largest Area/Aspect ratio) on the top routing layer are created as bump ports. A CLASS BUMP pin can be used either for power or ground nets. This option is enabled only when you process a pad cell in the *IO* bin and is switched off by default. A power net can have both CLASS CORE and CLASS BUMP ports.

Blockage

The following table describes the fields available on the *Blockage* tab of the Running step Abstract form.

Field	Description
Layer Assignment for Blockages	This section is used to specify how blockages are to be modeled in the abstract.

© 2023 Cadence Design Systems, Inc.

Field	Description
Layer	Specifies the layers on which you want to create blockages. You can do one of the following: ■ Select a layer from the drop-down list available from the technology file. After you have selected a layer from the drop-down list, it is removed from the list for the next layer row. ■ Type the layer name directly in the *Layer* column. Specify each layer only once. If you enter an invalid layer, Abstract Generator will issue a warning. If you do not create a table entry for a particular layer, Abstract Generator will not create any blockages on that layer in the final abstract.
Geometry Specification	Lets you enter the geometry specification for the layers on which you want the blockage to be implemented.

© 2023 Cadence Design Systems, Inc.

Field	Description
Blockage	Lets you specify the type of blockage you want to create on the specified layer: *Cover*, *Detailed*, or *Shrink*. The default is *Detailed* for the *Core* bin and *Cover* for the *Block* bin. Use the `AbstractBlockageUserDefinedWidth` option to set the effective width attribute for layer blockages. Spacing and effective width attributes cannot be set together. Effective width can be set only if the spacing value is set to zero. ■ *Cover:* A *Cover* blockage is a blockage that covers an entire block for each metal layer used by the block. This type of blockage is typically generated for blocks or IO cells where you want to prevent over-the-cell routing to improve performance or to avoid electrical effects between tracks. Cover blockages follow the overlap layer, if present; else they follow the PR boundary. The cover blockage model does not generate any blockages outside PR boundary. Only exceptions to this are the shapes that are affected by pin cutout regions, where these shapes are considered to be part of the layout. ■ *Shrink:* A *Shrink* blockage is a blockage that fills in the smaller, less useful free spaces in a block and leaves out the larger spaces. Based on the specified *Shrink Dist* or *Shrink Track*, Abstract Generator combines blockages within that distance of each other, filling the space between them with more blockage. This results in a small number of large blockages separated by spaces that a router can use. This allows over-the-cell routing without modeling each obstruction individually. The *Shrink* blockage generation slows down the overall process of abstract generation but ultimately increases the speed of routing significantly, which is better than when *Detailed* blockages are routed. ■ *Detailed*: A *Detailed* blockage is a blockage that covers geometries that are not part of pins on the routing layers, thereby allowing maximum routing within the block. This type of blockage is used for standard cells and possibly for higher-level routing layers in blocks. Do not create detailed blockages on contact or via layers for large blocks. Creating detailed blockages can adversely affect the performance of Abstract Generator because of the number of individual shapes that need to be processed. Consider creating *Shrink* type of blockage by setting a distance to merge most of the obstructions. You can speed up the detailed blockage creation in some scenarios.

© 2023 Cadence Design Systems, Inc.

Field	Description
Pin Cutout	Enables Abstract Generator to cut the blockage around pins for allowing a router to access the pins. See Impact of Cutout and Spacing Options on Blockage Types.
Max Space	Enables Abstract Generator to create a boundary blockage around a block such that the top-level routes are maximum possible distance away from the block. The boundary blockage is interspersed between pins along the block boundary. You can use this option when you want to do routing for cells even when their detailed layout is still not complete but the pin positions and boundary have been finalized. This option works only for *Cover* blockages and is not applicable for *Detailed* and *Shrink* blockages. See Impact of Cutout and Spacing Options on Blockage Types. The *Max Space* option is effective only when the *Pin Cutout* option is selected. Further, when the *Max Space* option is selected along with the *Pin Cutout* option, the *Corridor Cut* option is ineffective.
Corridor Cut	Enables Abstract Generator to create corridor paths that can allow routers to access internal pins. This option, when selected, first determines the existence of straight line paths that can route `minWidth` wires from the internal pins to the block boundary and then creates the corridor cuts. Also, Abstract Generator reads the direction of the pin and checks the possibility of creating corridor cut in a direction. If the pin direction matches with the nearest edge then the corridor is cut in that direction. However, if the pin direction does not match with the direction of the nearest edge then the corridor will not be cut and an error message will be displayed. See Impact of Cutout and Spacing Options on Blockage Types. You can use this option only when the *Pin Cutout* option is selected.
Cut Same	Specifies the distance to cut around pins on the same layer. If set as `0`, the blockage completely covers the design, including the pin shape, and there is no pin cut out. By default, Abstract Generator cuts out enough space to enable a via to be dropped from the layer above anywhere onto the pin without violation. This setting overrides the default size of the cut-out made around pins. The default is still used for any unlisted blockage layers. See Automatic Pin Stretching.

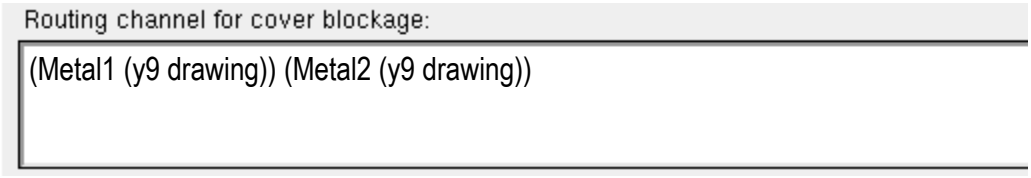
© 2023 Cadence Design Systems, Inc.

Field	Description
Cut Below	Specifies the distance to cut around pins on the layer below. By default, Abstract Generator cuts a window from a blockage layer that overlaps pins on the layer below, leaving enough space to enable a via to be dropped from the blockage layer anywhere onto the pin without violation. For example, if you are creating cover blockages for metal 4, then pin cutouts are created on metal 4 blockages for pins in both, metal 4 and metal 3. By default, the *Cut Below* field does not have any value. However, you can specify any value in this field. If you specify the value 0, then pin cutouts are created only on the current layer. In the above example, pin cutouts will only be created on metal 4. See Automatic Pin Stretching.
Shrink Dist	Specifies a minimum distance (in microns) between blockages. Blockages separated by a distance less than or equal to the specified distance are merged, irrespective of whether Pin Cutout is ON or OFF. Values entered in this field are considered only if you select to generate *Shrink* blockages.
Shrink Track	Specifies the minimum number of tracks between blockages. Blockages separated by a distance less than or equal to the specified number of tracks are merged. Values entered in this field are considered only if you select to create *Shrink* blockage. The distance, D, required to accommodate a given number of tracks, n, is calculated (in microns) using the following formula: $D = (n * \text{minTrackWidth}) + ((n + 1) * \text{minSpacing})$ If you specify both *Shrink Track* and *Shrink Dist*, the values are added and blockages separated by a distance less than or equal to the total distance are merged. If you do not supply a distance, Abstract Generator uses a default value of 2 tracks. If values are entered for layers for which *Shrink* blockage is not being generated, the values are ignored.

© 2023 Cadence Design Systems, Inc.

Field	Description
Boundary	Specifies the minimum distance between a *Cover* blockage and the cell boundary. The default value of this option is `0`. This option is valid only for the *Cover* blockage. The *Add*, *Edit*, and *Delete* buttons and the Specifying General Options in G-Spec Builder Form in conjunction with the Operators are used to work with the *Layer Assignment for Blockages* table. When you add a new layer row to the table, the type of blockage is set as *Detailed* for the *Core* bin and *Cover* for all other bins by default.
Cut window around pins large enough to drop via	Controls the size of the window cut into a blockage around a pin. This option is selected by default. When this option is selected, Abstract Generator cuts a window large enough for a via to be dropped to that pin. If you do not select this option, Abstract Generator cuts a window around the pin based purely on the range-based spacing rule for the pin width. Alternatively, use the `BlockageCutVia` option. Abstract Generator uses the `minSpacing` value for the minimum spacing for both metal and via layers. It does not use the `viaSpacing` rule. If you have `viaSpacing` tables, you need to ensure that the spacing for zero adjacent cuts is defined in the `minSpacing` rule. This option applies only to the layer on which the pin and blockage exist. However, for a given blockage layer, Abstract Generator additionally cuts around pins on the layer below, again leaving space for a via to be dropped without violation. This option has no effect on the size of the cut-out around pins on the layer below. If you want the cut-outs on all layers to be based only on the pin width, set the *Cut Below* value to `0` for the given blockage layer in the *Layer Assignment for Blockages* table.

© 2023 Cadence Design Systems, Inc.

Field	Description
Routing channel for cover blockage	Creates a routing channel when generating a cover blockage. Consider an example below:  Here, *Metal1* refers to layer through which a routing obstruction needs to be cut. *(y9 drawing)* is the layer-purpose pair that indicates the shape of the routing channel. You can use the `AbstractBlockageRoutingChannelOnLayers` option to create a routing channel when generating a cover blockage. **Example** `absSetBinOption("Block" "AbstractBlockageRoutingChannelOnLayers" "(Metal1 (y9 drawing)) (Metal2 (y9 drawing))")` Layers `(y [0-9] drawing)` are system-defined layers. So, the results may not be accurate when these layers are used for creating routing channels.

Density

The following table describes the fields available on the *Density* tab of the Running step Abstract form.

Field	Description
Calculate metal density	Specifies that metal density information is to be generated for the selected cell(s). When this option is selected, Abstract Generator generates the fill blockages that are specified in the DEF file. During LEF generation, these fill blockages are exported out as density. By default, this option on the *Density* tab is enabled. The other options become active only when the *Calculate metal density* option is enabled.

© 2023 Cadence Design Systems, Inc.

Field	Description
Use layer assignment for signal extraction, *Use layer assignment for antenna extraction*, *Use layer assignment for power extraction*	Lets you specify metal layers and their geometry specifications for metal density calculation. You can use the same metal layers and their corresponding geometry specifications as you would have specified for signal nets, power nets, or antenna data extraction in the Extract step. To do this, select one of the following options: ■ *Use layer assignment for signal extraction* ■ *Use layer assignment for antenna extraction* ■ *Use layer assignment for power extraction* These options are mutually exclusive and selecting one disables the other two options. Only metal layers specified in the selected table will be considered for metal density calculation.
Layer Assignment for Metal Density Regions	This section provides options to select different set of geometry specifications for metal layers. To use the geometry specifications from this table, ensure that none of the three layer assignment options is selected. You can turn off these options individually. If you select to use a layer assignment table for density calculations from a different tab, you can still use the *Width* and *Height* columns of the *Layer Assignment for Metal Density Regions* table on the *Density* tab. In this case, though the geometry specifications in the *Layer Assignment for Metal Density Regions* table will not be considered for metal density calculation but you need to ensure that a valid geometry specification exists for every layer specified in this table.
Layer	Lets you remove or edit the existing layers or add any removed layers. Density data will not be calculated for metal layers that are not specified in the table. This column is pre-populated with the metal layers from the technology file. If you use the *Layer Assignment for Metal Density Regions* table and you have specified a non-metal layer in the table, Abstract Generator will issue a warning message during density calculation.
Geometry Specification	Lets you select the layer names corresponding to each metal layer name in the *Layer* column. You can modify these values as required. The *Width* and *Height* columns are described in the section, Specifying the Dimensions of the Density Window.
Width	Lets you specify the width of the density window either for all the layers or separately for each layer.

© 2023 Cadence Design Systems, Inc.

Field	Description
Height	Lets you specify the height of the density window either for all the layers or separately for each layer.
Default density window width	Lets you specify the same width value for all the layers.
Default density window height	Lets you specify the same height value for all the layers. The default value for both the *Default density window width* and *Default density window height* fields is `20`, which is derived by calculating 10% of a typical `130nm` window and 20% of a typical `90nm` density check window size. You can modify this value to get Abstract Generator to override the default window size without having to change the *Width* and *Height* values for every layer.

Fracture

The following table describes the fields available on the *Fracture* tab of the Running step Abstract form.

Field	Description
Fracture	This section provides options to specify whether you want to fracture pins or blockages by using the options in the *Fracture* section - *Fracture pins* and *Fracture blockages*. By default, both these options are disabled.

© 2023 Cadence Design Systems, Inc.

Field	Description
Fracture pins	Controls whether Abstract Generator creates each pin shape as a set of maximum rectangles or as a polygon. This lets you model 45-degree pins if your design requires it. This option is available for all bins. When this option is not selected, which is the default, Abstract Generator creates each blockage shape as a polygon. This setting is recommended if the abstracts are to be routed using the IC Shape-Based Router. If you select this option, Abstract Generator creates each pin shape as a set of maximum rectangles. You can also choose to fracture only rectilinear pin shapes. To do this, you can use the `AbstractFractureRectilinearShapes` option of the `absAbstract` SKILL function. `absSetBinOption("Block" "AbstractFractureRectilinearShapes" "true")` If you are using the *Create boundary pins* or *Create ring pins* option on the *Adjust* tab and Abstract Generator has stretched a pin, then that pin is not fractured by this option. For example, if you have boundary pins for power nets but not for signal nets, only the power nets are fractured.

Fracture blockages	Controls whether Abstract Generator creates each blockage shape as a set of maximum rectangles or as a polygon. This option is available for all bins. If you select this option, Abstract Generator creates each blockage shape as a set of maximum rectangles. When this option is not selected, which is the default, Abstract Generator creates each blockage shape as a polygon. This setting is recommended if the abstracts are to be routed using the IC Shape-Based Router.
	
	During blockage generation, Abstract Generator converts ellipses, circles, and donuts into polygons before fracturing the blockages. By default, 64-point polygons are generated, which are snapped to the nearest manufacturing grid. You can overwrite this setting by using the following:
	
	`absSetOptionabsSetOption("numOfPointsInEllipseOrDonut" "number_of_points")`
	
	Example:
	
	`absSetOption("numOfPointsInEllipseOrDonut" "64")`

45-degree geometry	This section provides options to convert a 45-degree geometry.

© 2023 Cadence Design Systems, Inc.

Field	Description
Stair-step coverage	Controls the degree to which stair-step blockages cover the 45-degree geometry. There are three settings available for this option: *full*, *truncate*, and *partial*.
Stair-steps width	Specifies the width of the blockage geometry to be created. Enter a value greater than zero. There is no minimum width restriction. The default value is half the `minSpacing` of the `Metal1` layer. If the `minSpacing` of `Metal1` layer is not found in the technology data, then the default value is set to `5`.

Site

The following table describes the fields available on the *Site* tab in the Running step Abstract form.

Field	Description
Site name	Lets you select a site name defined in the technology library from the available list to be applied to all cells in the current bin. Alternatively, select the *blank* option and allow Abstract Generator to select an appropriate site based on cell property settings. The name of the site will appear in the LEF file exported from Abstract Generator. For any site to appear in the LEF generated from Abstract Generator, it must be defined in the technology library.
Define new site name	Lets you specify the name of a new site to be used by all cells in the current bin. If the site does not already exist, Abstract Generator defines it in the technology file. An entry in the *Define new site name* field overrides any setting in the *Site name* field. Leaving this field blank automatically instructs Abstract Generator to use the site selected in the *Site name* field. Both *Site name* and *Define new site name* fields associate cells with a single site.

© 2023 Cadence Design Systems, Inc.

Field	Description
Calculate site pattern for gate-array cells	Calculates site pattern for gate-array standard cells. For LEF 5.8, Abstract Generator can derive site patterns from the layout of gate-array standard cells. Abstract Generator provides this support only for gate-array cells with a single level of hierarchy. This site pattern calculation feature is available only for the *Core* bin. For valid site pattern calculation, the selected sub-cellviews (building blocks of gate-array cells) should be either associated to a predefined site or mapped to an already existing site. You can map the sub-cellviews to sites in the *Sub-Cellview to Site Mapping* table. This option works exclusive of the single site creation fields on the *Site* tab.
Sub-Cellview to Site Mapping	Lets you map the sub-cellviews to sites. For valid site pattern calculation, the selected sub-cellviews (building blocks of gate-array cells) should be either associated to a predefined site or mapped to an already existing site.

Overlap

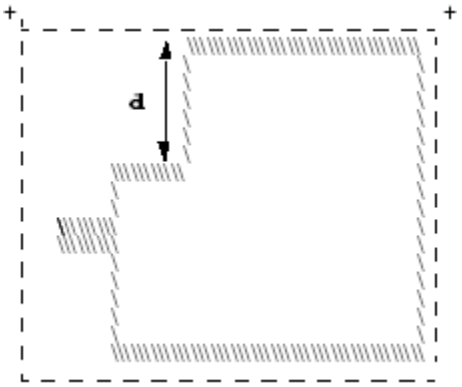
The following table describes the fields available on the *Overlap* tab in the Running step Abstract form.

Field	Description
Create overlap boundary	Lets you specify whether Abstract Generator should create an overlap boundary. This option is a cyclic field and you can choose one of the following values: ■ *always*: Abstract Generator always creates an overlap boundary. Any existing overlap layer geometry is overwritten. ■ *as needed*: Abstract Generator creates an overlap boundary if it is not already present or if it is present and was previously calculated by Abstract Generator. ■ *off*: Abstract Generator never creates an overlap boundary.

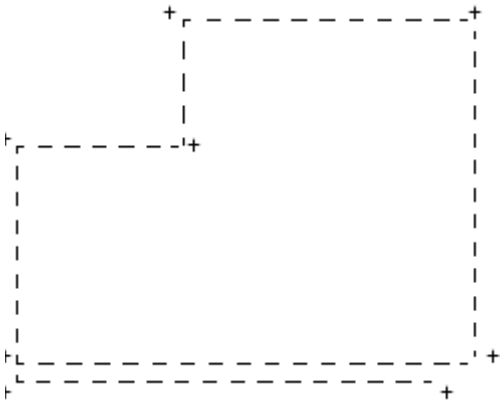
© 2023 Cadence Design Systems, Inc.

Field	Description
Using geometry on layers	Lets you specify the layers to be used in the overlap boundary calculation. The boundary is drawn so that it encloses all the geometry found on these layers. If you specify only a layer name instead of a layer-purpose pair, Abstract Generator considers all the purposes on that layer except the ones specified in the *Ignored Purpose(s)* field on the *General* tab (*File – General Options*).
Size factor to apply	Lets you specify (in microns) the value by which an overlap layer is to be increased. This option is enabled when the *Create overlap boundary* option is set to *always* or *as needed*. The default values depend on the current bin. The default for the *Block* bin is `0.0`; the default for the *Core* bin is half the minimum separation defined for the `metal1` layer. You can use this option to ensure that the placement of cells with abutting overlap layers does not lead to geometry contained in these cells causing violations. You are most likely to need this option when computing the overlap layer for standard cells that are placed automatically and abutted in rows. Blocks are abutted only rarely and should not require you to use this option.

© 2023 Cadence Design Systems, Inc.

Field	Description
Smooth factor to apply	Controls the shape of the overlap layer that is produced. This option is enabled when the *Create overlap boundary* option is set to *always* or *as needed*. The default values depend on the current bin. The default for the *Block* bin is `100` microns; the default for the *Core* bin is `0`. You can use this option to create less complex overlap boundary shapes, typically when processing blocks. The value you supply is the maximum distance that you do not want to be cut out of the overlap layer. Abstract Generator takes the computed overlap boundary and removes any cut-outs with a maximum dimension less than the specified distance. Consider the following block. The PR boundary has been indicated with a broken line (`--`), and the outline of the geometry has been shown using the backslash character (`\\\`). This area marked by the backslashes indicates the area of the cell that contains shapes on the layers that will be used to create the overlap layer. (This area would become the overlap layer if the *Smooth factor to apply* was set to `0`). 

© 2023 Cadence Design Systems, Inc.

Field	Description
	When computing the overlap layer, it is likely that you want to retain the cut-out in the top left corner as free space. To do this, you must specify the value for the *Smooth factor to apply* option as `d`, where `d` is the distance in microns slightly less than the minimum dimension of the cut-out. Abstract Generator removes all other free space between the initial overlap boundary and the PR boundary but keeps the region in question. 

Related Topics

Customizing Pin Shapes in Standalone Abstract Generator

Modifying Blockage Geometry in the Abstract View

Density Calculation

Fracturing Pins and Blockages

Specifying Sites in Standalone Abstract Generator

Creating an Overlap Boundary

© 2023 Cadence Design Systems, Inc.

Running Step Extract Form

Use the Running step Extract form to extract geometries on signal or power nets or to calculate process antenna information.

This form contains the following tabs.

Tab	Description
Signal Tab	Allows you to view and modify the options for extracting signal nets. You can specify the layers through which you want Abstract Generator to extract, the layers on which pins are to be created, the limits of extraction, and the way in which must-join relationships are to be handled.
Power Tab	Allows you to view and modify the options for extracting power nets. You can specify the layers through which you want Abstract Generator to extract, the layers on which pins are to be created, and the limits of extraction.
Antenna Tab	Allows you create process antenna information for custom blocks and standard cells
General Tab	Allows you to use the existing connectivity information in your design to create nets, to specify the connectivity between layers, and to have a control over final pin geometry.

© 2023 Cadence Design Systems, Inc.

Signal Tab

The following table describes the fields available on the *Signal* tab of the Running Step Extract form.

Field	Description
Extract signal nets	Runs connectivity extraction for signal nets. Therefore, when this option is turned on, Abstract Generator extracts all the pin shapes created during the Pins step. When the option is turned off, Abstract Generator copies the pin geometry generated in the Pins step but the pin shapes now have purpose *net* rather than purpose *pin*. This is the default while processing blocks. Selecting the *Extract signal nets* option activates the Layer Assignment for Signal Extraction table on the *Signal* tab. In this table, you specify the layers through which you want Abstract Generator to extract. If a pin-short is detected, Abstract Generator displays an error message detailing the cause of the problem. The error might occur as a result of either a bad layer, bad extraction option settings, or perhaps a short between “false” boundary purpose shapes and other real shapes.
Layer Assignment for Signal Extraction	This section lets you specify the geometry specification details in the respective layer assignment table. For example, you can restrict `metal1` to be just `metal1` purpose *pin* or you can specify `metal1` to be equal to shapes only where `metal1` purpose *pin* and `metal1` purpose *net* overlap.
Layer	Lets you select from a list of all the layers specified in the technology file through which you want to extract signal or power nets or antenna data. Each row in this column provides a drop-down list of the layers in the technology file. You can use this column to limit the extraction to shapes on a subset of all the layers. For example, if you want to prevent Abstract Generator from extracting through via layers, you can delete those layers from this list. You can create pins corresponding to the layers specified in the *Layers* column.

© 2023 Cadence Design Systems, Inc.

Field	Description
Geometry Specification	Allows you to enter a layer-geometry specification for each layer included in the *Layer* column of the table that you want to use for extraction. For example, if you want to extract through diffusion layers you will usually want to pre-process the diffusion geometry by subtracting the polysilicon layer geometry first. You can achieve this by specifying the following geometry specification for the diffusion layer: `diff andnot polysilicon` The first entry in the expression is the layer name. The second entry is a more complex expression combining as many layer names and layer operators as required. If you specify only a layer name instead of a layer-purpose pair in the *Geometry Specification* column, Abstract Generator considers all the purposes on that layer except the *Ignored Purpose(s)*. Use the G-Spec Builder Form to help you with the geometry specification. To access this form, place the cursor in the *Geometry Specification* field for the required layer and click the *Edit* button. You can also use this column to specify more than just the list of layers to be extracted. Rather than listing just the layer name for each layer, you can enter an expression that is used to specify how the input geometry for the extraction of that layer is derived. If you attempt to define two geometry specifications for the same layer, an error will be issued.

© 2023 Cadence Design Systems, Inc.

Field	Description
Connectivity	Lets you can specify the type of connectivity setting, *Strong* or *Weak*, for a particular layer. The connectivity setting is valid for a layer only if a geometry specification is provided for that layer. When extracting signal nets, you can set certain layers as "weak" layers and instruct the extractor to extract through these as well. The shapes extracted from weak layers will be stored in the LEF file as `weakly-connected`. The router handles strongly and weakly connected pins differently. Where a LEF `PIN` has two strongly-connected shapes, the router can connect to one of the shapes to form the connection to that pin, then continue routing from the other shape on the same pin without adversely affecting performance. However, when the shapes are weakly-connected, it adversely affects the delay of the net. Weakly-connected shapes are written out to the LEF file in separate `PORTS` of the same LEF `PIN`. When a router encounters two weakly-connected shapes within a LEF `PIN`, it does not feedthrough from one shape onto the other; it routes to only one of the shapes. You would typically state that a layer has weak connectivity if it has high resistivity, as is the case with polysilicon layers.
Create Pins	Lets you select the layer on which you want pins to be created as a subset of the layers used in the extraction. This helps to determine the shapes found by the extractor that are converted into pins. By default, this option is active when a new layer is added to the table.
Add	Adds a new layer row to the table.
Edit	Opens the G-Spec Builder Form for the selected layer.
Delete	Deletes the selected layer rows. Check the box to the left of a row to mark it for deletion.
Extract Limitations	This section lets you speed up the extraction process by limiting how far from the starting shape and how far down the hierarchy should Abstract Generator search for pins.

© 2023 Cadence Design Systems, Inc.

Field	Description
Maximum depth	Lets you specify how far down the instance hierarchy should Abstract Generator search for shapes for extraction. The default value for this field is `32`, which is the maximum depth of hierarchy levels in the database. Limiting the hierarchy search depth is useful if you know the likely location of the shapes to be extracted. For example, if you know that all the geometry that needs to be converted into pins is at the top level (in a block most likely), you can set the value of the *Maximum depth* field to `0`. This value will tell Abstract Generator to look for shapes only at the top level of the hierarchy and not search further, in turn, enhancing the speed of the Extract step. This might be the case with signal pin geometry that might exist on different levels of the design hierarchy, but is useful for connecting to only at the top level where it would typically be at its widest. You may want to restrict the level depth to extract through to prevent pins from being created on lower levels of the instance hierarchy. However, if you specify a lower value for the *Maximum depth* field and have also selected to perform antenna extraction, extraction will be performed twice, first for signal nets and then for antenna data. See Specifying Layers and Geometries for Extraction.
Maximum distance	Allows you to restrict the shapes considered by the extractor to those that intersect a bounding box around each starting pin shape. The extractor will only extract shapes that intersect the bounding box. The size of this bounding box is specified in microns in the *Maximum distance* field. This specification can help to reduce the extraction time for large blocks, where you want to restrict the pin geometry to be near the block boundary. For example, when extracting signal or power nets in a large block, you would supply a maximum extract distance to prevent extracting the entire signal/ground net or power/ground net within the block.

© 2023 Cadence Design Systems, Inc.

Field	Description
Minimum width	Specifies the minimum width of the shapes to be extracted. The extractor extracts only shapes that are greater than or equal to the width you specify. The minimum width value is applied to all layers except cut layers. The minimum width criteria is *not* applied to cut layers because the extractor would always stop at the narrow cut shapes. The benefit of using this option is that it can significantly reduce the time taken to extract useful geometry from large nets that have a number of routing widths. For example, if you are interested in extracting ring pin geometry from a block that has internal power rings, you can specify the minimum width for extracting power nets to be equal to the width of the ring pins. The extractor would then only extract the ring pin geometry.

© 2023 Cadence Design Systems, Inc.

Field	Description
Create Must Join Pins	This section controls what happens when the extractor finds two disjoint groups of shapes extracted for the same net. In the Pins step, where the same text appears in more than one label, multiple pins are created for the same net. These pins are the starting points for the connectivity extractor. For example, if a cell has a net named `A`, the layout designer might label the net `A` at two different locations in the cell. Abstract Generator will create two starting pins for net `A` and the extractor will extract from both of these starting pins. The extractor might find that the geometries are connected inside the cell. Sometimes, in such cases, the geometries extracted from the two starting pin shapes remain disjoint. This might happen due to one of the following reasons: ■ The pins are connected inside the cell but the extraction performed by Abstract Generator has not found that connectivity. This can happen when you limit the extractor to certain layers or when you specify a *Maximum distance* for extraction or limit the hierarchy search to a *Maximum depth*. ■ The pins are disjoint inside the cell, and the layout designer intended that the router should connect to both pins in order to correctly connect net `A`. In LEF, this set of pins are termed as “must-join”.s If the `LEF54_MUSTCONNECTPORTS` property is set on any terminal in the layout, it will be reflected in the abstract view.
Always	Allows the Abstract Generator to establish a must-join relationship between all disjoint groups of geometries extracted from separate starting pins on the same net. A must-join relationship is represented in LEF as must-join terminals. Abstract Generator creates must-join terminals with the same name as the original terminal, but also includes the suffix `_mj`***xxx*** where ***xxx*** is the number of the must-join group used internally to identify all the shapes in the same group. For example, in the scenario described above, Abstract Generator would create a must-join terminal called `A_mj1`.

© 2023 Cadence Design Systems, Inc.

Field	Description
Only on terminals named	Lets you control when to create LEF must-join relationships between disjoint geometries extracted from separate starting pins on the same net. You can enter a list of terminal names, each separated by a single space. If the extractor finds a terminal name matching one of the names supplied, and if there are disjoint geometries from different starting pins on that net after extraction, Abstract Generator creates a `MUSTJOIN` relationship for that terminal. This might be useful when you know the terminals expected to have must-joins. Specify their names in this field, and if the extraction process creates other disjoint groups of geometries, they will not be turned into must-join groups and will instead be treated as strongly connected. Abstract Generator informs you when it creates must-join terminals as the result of an extraction. It also reports when it detects disjoint geometry groups but the option settings did not allow the creation of the must-join terminal. If there are must-join terminals already defined in the layout, they will be retained in the final abstract irrespective of whether you use the options in the *Create Must Join Pins If Required* section or extract signal nets. Abstract Generator would propagate the must-join terminals only if the pin type (input, output, inout) of the must-join pins is the same; otherwise, Abstract Generator will issue an appropriate error message.

© 2023 Cadence Design Systems, Inc.

Power Tab

The following table describes the fields available on the *Power* tab of the Running Step Extract form.

Field	Description
Extract power nets	Directs Abstract Generator to run connectivity extraction for power nets. Therefore, when this option is turned on, Abstract Generator extracts all the pin shapes created during the Pins step. This option is on by default for standard cells. When the option is turned off, Abstract Generator copies the pin geometry generated in the Pins step but the pin shapes now have purpose *net* rather than purpose *pin*. This is the default while processing blocks. **Note:** When processing a custom block in which ring style power pins are required, it is necessary to keep the *Extract power nets* option on. This will ensure that enough power geometry is extracted that can be processed by the ring pin identification algorithm in the Abstract step to identify the shapes to be turned into ring pins. Selecting the *Extract power nets* option activates the *Layer Assignment for Power Extraction* table on the *Power* tab. In this table, you specify the layers through which you want Abstract Generator to extract. Cadence recommends that you should not use the *Extract power nets* option for blocks or IO cells unless you are sure that you need to; for example, use this option only if you want to create ring pins. If you do need to extract power nets, do not create pins on a given layer unless it is necessary. For example, do not create pins on polysilicon, diffusion or contact layers, except where there are dominant cut-layer spacings. You should also turn on the *Create ring pins* or *Create boundary pins* options in the Abstract step to ensure that not all of the extracted geometry is turned into pins. If you do not, Abstract Generator must cut around each pin, which would increase the time taken to generate the blockage model.

© 2023 Cadence Design Systems, Inc.

Field	Description
Layer Assignment for Power Extraction table	Specifies the geometry or shapes that should be present on a particular layer when power net extraction is performed. The entries in the *Layer Assignment for Power Extraction* table are used by other functions even when the *Extract power nets* option is switched off. The situations in which this might happen are discussed below. ■ In the Extract step, if you have a geometry specification with more than two parts, for example as in, `(metal1 andnot marker)` Abstract Generator finds shapes that touch or overlap the pin shapes. This can result in the creation of pins that are different from the ones created in the Pins step. In the above geometry specification example, it might happen that the design has two nets shorted together on `metal1` and the layout has a `marker` layer that indicates the boundary between the two nets. Therefore, any shape on the `metal1` layer and not the `marker` layer which touches an original pin is turned into a pin. ■ In the Abstract step, Abstract Generator tries to stretch the pins. To do this, it uses the geometry specification entered in the table to ensure that it converts only the shapes that satisfy the specification into pins. See Relationship between Signal and Antenna Tabs.
Create Must Join Pins If Required	Lets you control what happens when the extractor finds two disjoint groups of shapes extracted for the same net.

© 2023 Cadence Design Systems, Inc.

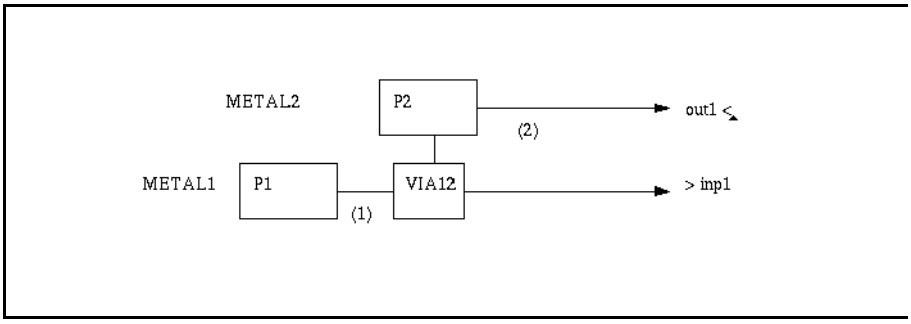
Antenna Tab

Field	Description
Calculate hierarchical antenna	Calculates antenna values hierarchically by summing the antenna data on standard abstracts. By default, this option is switched off and disabled. It becomes active only when one of the other antenna calculation options is selected. This option bypasses the check for *Gate* or *Drain* regions in the *Layer Assignment for Antenna Regions* table because these measurements are not required for hierarchical antenna calculations. If this option is selected and there are no abstract instances, Abstract Generator performs a flat antenna calculation instead. If this is the case and the *Gate* and *Drain* regions are not specified, Abstract Generator warns you that neither process antenna information has been calculated nor the diffusion and polysilicon geometries have been extracted. **Calculation** The following equation is the `MaxAreaCar`, `MaxSideAreaCar` and `MaxCutCar` hierarchical antenna calculation for the abstract instance terminal on a particular net. This is maximized with the `MaxCar`'s for the other gate nodes to get the overall max. `M_i CAR = M_i PAR + max{ M_i-1 CAR, (M_i MAX_CAR(H) - M_i PAR(H)) }` where, ■ `M_i` is the fabricated metal layer. ■ `M_i PAR` is the partial metal area divided by the gate area under the metal layer. ■ `M_i-1 CAR` is the cumulative area ratio calculated for the layer below. ■ `M_i MAX_CAR(H)` is the `pcsAntennaMaxAreaCAR` information from the instance terminal. ■ `M_i PAR(H)` is `pcsAntennaPartialMetalArea` or `pcsAntennaGateArea` from the instance terminal.

© 2023 Cadence Design Systems, Inc.

Field	Description
	If `pcsAntennaMaxAreaCAR` is not specified, the calculation takes a simpler form as in: `M_i CAR = M_i PAR + M_i-1 CAR` See Keywords Used in Antenna Calculation

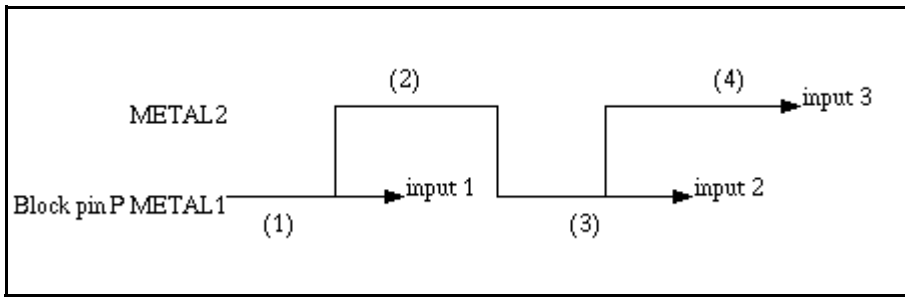
© 2023 Cadence Design Systems, Inc.

Field	Description
Calculate input pin antenna	Calculates either the ANTENNASIZE LEF (if you are using LEF 5.3) or the ANTENNAGATEAREA attribute (if you are using LEF 5.4 or above) for input pins. By default, this option is switched off for standard cells and custom blocks. The following example illustrates how the antenna values are calculated for inout pins. **Example** Consider the following standard cell output that has internal feedback to the gate inp1 and two pins (P) in the abstract P1 (on METAL1) and P2 (on METAL2).  ANTENNAGATEAREA METAL1 = Area of inp1 gate ANTENNAGATEAREA METAL2 = Area of inp1 gate ANTENNADRAINAREA METAL2 = Area of out1 drain ANTENNAPARTIALMETALAREA METAL1 = Area of METAL1 segment (1) - area of pin P1 ANTENNAPARTIALMETALAREA METAL2 = Area of METAL2 segment (2) - area of pin P2 ANTENNAPARTIALMETALSIDEAREA METAL1 = Perimeter of (segment (1) - P1) + METAL1 thickness ANTENNAPARTIALMETALSIDEAREA METAL2 = Perimeter of (segment (2) - P2) + METAL2 thickness See Keywords Used in Antenna Calculation.

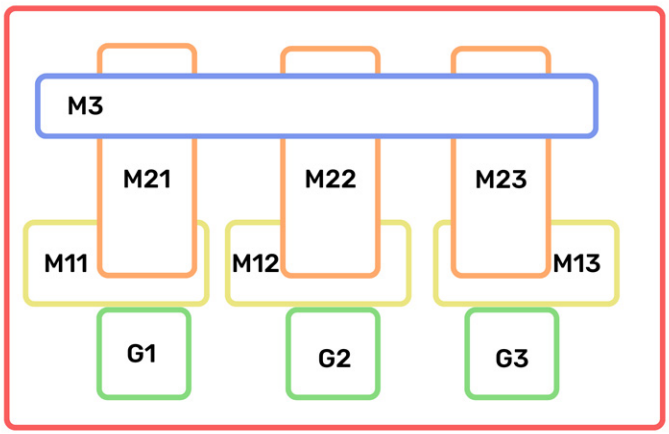
© 2023 Cadence Design Systems, Inc.

Field	Description
Calculate output pin antenna	Calculates either the `ANTENNASIZE` LEF (if you are using LEF 5.3) or the `ANTENNAGATEAREA and ANTENNADIFFAREA` attributes (if you are using LEF 5.4 or above) for output pins and tristate pins. By default, this option is switched off for standard cells and custom blocks.
Calculate inout pin antenna	Calculates either the `ANTENNASIZE` LEF (if you are using LEF 5.3) or the `ANTENNAGATEAREA and ANTENNADIFFAREA` attributes (if you are using LEF 5.4 or above) for IO pins and tristate pins. By default, this option is switched off for standard cells and custom blocks.

© 2023 Cadence Design Systems, Inc.

Field	Description
Calculate antenna metal area	Calculates the area of metal that would be connected to the pin when that layer is fabricated. This option is designed to be used with blocks. The result is expressed in square microns. By default, this option is switched off and disabled. It becomes active only when the non-side area ratio is specified in the technology file. The calculated antenna metal area is added by the router to the routing metal at the next level up in order to compute whether or not there is an antenna violation on that pin or on any other pin on the same net. Use the `ExtractAntennaIncludePolyCAR` option to include poly shape area when calculating the antenna metal area. When set to `true`, poly shapes are considered as metal shapes and are therefore included in the calculation of `ANTENNAPARTIALMETALAREA`. Consider the following example that describes the calculation of `ANTENNAPARTIALMETALAREA`. **Example** For each metal layer, you must calculate the area of metal attached to the pin that will be connected when that layer is fabricated, that is, the metal on the same external pin side of some higher level piece of metal. You need to calculate the metal area only for those layers where there is pin geometry and any higher layers used internally on the connection to the gate. 

© 2023 Cadence Design Systems, Inc.

Field	Description
	From the above figure, `ANTENNAGATEAREA METAL1` = Area of gate (`input1`) `ANTENNAGATEAREA METAL2` = Area of gate (`input1`) + area of gate (`input2`) + area of gate (`input3`) `ANTENNAPARTIALMETALAREA METAL1` = Area of `METAL1` segment (1) - area of `Block pin P` `ANTENNAPARTIALMETALAREA METAL2` = Area of `METAL2` segment (2) + area of `METAL2` segment (4) `ANTENNAPARTIALMETALSIDEAREA METAL1` = Perimeter of (`METAL1` segment (1) - `Block pin P`) x `METAL1` thickness `ANTENNAPARTIALMETALSIDEAREA METAL2` = (perimeter of `METAL2` segment (2) + perimeter of `METAL2` segment (4)) x `METAL2` thickness See Keywords Used in Antenna Calculation To enable Abstract Generator to consider only those paths that contain pins and not all the shapes and paths that are part of terminal, you can use the ExtractAntennaPinPathsOnly { true | false } SKILL option. Consider two pins, `M11` and `M13`, on `METAL1` layer. If `ExtractAntennaPinPathsOnly` is set to `true` and `ExtractSig` is set to `false`, Abstract Generator considers only those paths that contain pins. 

© 2023 Cadence Design Systems, Inc.

Field	Description
	For example, gateArea `M1` and `M2` = `Min(G1,G3)` Otherwise, Abstract Generator considers all the paths, that is, gateArea `M1` and `M2` = `Min(G1,G2,G3)`
Calculate antenna metal side area	Calculates the side area of the metal (perimeter x thickness) that would be connected to the pins when that layer is fabricated. By default, this option is switched off and disabled. It becomes active only when the side area ratio is specified in the technology file.
Layer Assignment for Antenna Regions	This section lets you assign layers to antenna regions for identifying the gate and drain regions and to enable antenna calculation. This field is enabled when at least one of the calculate antenna options is active. If the information in the technology file is set up correctly (layers need to be defined in the `functions` section of the `LayerRules` section), Abstract Generator automatically adds the entries to the table. In LEF, polysilicon and diffusion layers are defined as masterslice layers and both are converted to poly layers after the LEF data is imported to a technology file. Distinction between these layers is required for antenna calculation. So, Abstract Generator searches for the layer names `oxide`, `diff`, `ndiff`, `diffn`, `pdif`, `pdiff`, and `diffp` corresponding to the layer type `poly`, marks those layers as diffusion layers in the technology file, and removes them from the `poly` layer list. It then uses this information to automatically populate the *Gate* and *Drain* regions in the *Layer Assignment for Antenna Regions* table.
Layer	Lets you select the polysilicon and diffusion layers specified in the technology file.

© 2023 Cadence Design Systems, Inc.

Field	Description
Geometry Specification	Lets you enter a layer-geometry specification defining the gate or drain geometry regions. For example, ■ `(polysilicon and (pdiff or ndiff)` defines a gate region ■ `(pdiff (pdiff andnot polysilicon)` defines a drain region Use the Specifying General Options in G-Spec Builder Form to help you with the geometry specification. To access this form, place the cursor in the *Geometry Specification* field for the required layer and click the *Edit* button. You can define only one geometry specification for each layer.

Region	Specifies whether the layer is a *Gate* or *Drain* region.
	■ *Gate:* When calculating the size of a MOS transistor gate by using the option *Calculate input pin antenna*, Abstract Generator must measure the area of the region where polysilicon overlaps diffusion rather than the area of the polysilicon wire itself. Consequently, a geometric operation (`AND`) must be carried out to find the gate geometry. Selecting *Gate* as the region and entering the following in the *Geometry Specification* field specifies this operation:
	`(poly (poly and (ndiff or pdiff))`
	■ *Drain:* When using either the *Calculate output pin antenna* or *Calculate inout pin antenna* option, Abstract Generator cannot consider only the area of the drawn diffusion; it needs to remove the drain area from the area of the diffusion. The following geometry specification calculates the n-diffusion drain by subtracting the polysilicon from the n-diffusion geometry:
	`(diffn (diffn andnot poly))`
	The same principle applies when calculating p-diffusion.
	`(diffp (diffp andnot poly))`

© 2023 Cadence Design Systems, Inc.

Field	Description

Oxide	Lets you apply a unique oxide type for up to 32 diffusion layers. This option offers 32 oxide types (the maximum number handled by LEF): *Oxide1 … Oxide32*.
	
	If you list more than four diffusion layers, layers with the same *Oxide* type are treated in the same way. You can also select *Oxide1* for all the listed diffusion layers, in effect replicating the LEF 5.7 or above antenna style model.
	
	If the *Oxide1* keyword is associated with a particular diffusion layer, it means that the gate area over this diffusion area and the ANTENNAMAXAREACARs associated with this gate area use the *Oxide1* antenna ratios to determine antenna violations in Innovus.
	
	An example of LEF 5.5 could contain:
	
	`LAYER METAL1`
	
	`ANTENNAMODEL OXIDE1 ;`
	`ANTENNAAREARATIO 14 ;`
	`ANTENNAMODEL OXIDE2 ;`
	`ANTENNAAREARATIO 18 ;`
	
	`....`
	
	`END METAL1`
	
	and
	`MACRO mycell`
	
	`PIN mypin`
	
	`....`
	
	`ANTENNAMODEL OXIDE1 ;`
	`ANTENNAGATEAREA 8.2 LAYER M1 ;`
	`ANTENNAMODEL OXIDE2 ;`
	`ANTENNAGATEAREA 7.2 LAYER M1 ;`
	
	`....`
	
	`END mypin`
	
	`END mycell`

© 2023 Cadence Design Systems, Inc.

Field	Description
	The *Oxide* option is disabled if you select *Gate* as the *Region* type for a particular layer. Similarly, if you select an *Oxide* type for a layer, the *Region* type is changed to *Drain*. The default value " " corresponds to *Oxide1*.
Use different layer assignments for antenna calculations only	Lets you set layer assignments for antenna extraction different from those set for signal net extraction set on the Relationship between Signal and Antenna Tabs. Selecting this option activates the *Layer Assignment for Antenna Extraction* table and consequently over-rides any *Layer Assignment for Signal Extraction* option settings. A "behind-the-scenes" antenna extraction calculation can then be performed. If you are performing "behind-the-scenes" antenna calculations and using replay scripts, you must change the `extractLayersSig` option in the script to `antennaLayers` for it to run successfully. Selecting this option can be useful, for example, in such cases where you want to create pins from shapes which are touching the initial pin on the same layer and, if you had to specify the via layers to extract through, the Extract step may create pin shapes on other metal layers which you do not require. This, in turn, could cause electro-migration issues. By default, this option is disabled and off. It is activated if one or more calculate antenna options are selected. See Relationship between Signal and Antenna Tabs.
Layer Assignment for Antenna Extraction	This section is active when the *Use different layer assignments for antenna calculations only* option is selected.
Layer	Lets you specify the layer for which you want to extract antenna data.
Geometry Specification	Lets you enter a for each layer through which you want to extract antenna data.

© 2023 Cadence Design Systems, Inc.

Field	Description
Signal nets to be excluded from antenna calculation	Allows you to specify the nets that should not be considered for antenna extraction and therefore, lets you avoid antenna calculations for the specified pins. For example, by specifying the expression as `in*` for the signal nets to be excluded, you can exclude all nets with names beginning with `in` from antenna calculation. This field is enabled only if one of the calculate antenna options is selected.

© 2023 Cadence Design Systems, Inc.

General Tab

Field	Description
Use net information in design	Allows Abstract Generator to skip the extraction based on overlapping shapes and use the existing connectivity information in your design to create nets. Selecting this option for designs with existing connectivity information results in huge performance gain.
Layer connectivity	Allows you to specify the layer connectivity without having to add via definitions in the technology file. The default value is however based on the connectivity derived from analyzing the vias in the technology file, for example `(M1 M2 V12);` the `poly` and `diffusion` connections, for example `(POLY M1 CONT) (DIFF M1 CONT);` and `local interconnect` layers. The format for entering a value in this field is to list the layers as a number of entries; each entry being either a layer pair or layer triple. Example 1: `(layer1 layer2 layer3) (layer4 layer5 layer6)... (layer7 layer8)...` In the above example, a triple layer entry means that `layer1` connects to `layer2` through the via layer `layer3`. A pair means that the first layer, `layer7`, connects to the second layer, `layer8`, by overlap (for example, local interconnect to polysilicon). You must specify these layer connections in the *Layer Assignment for Signal Extraction* table. Example 2: `(Li1 Li2) (PO Li1) (OD Li2)` In the above example, layer connectivity is defined between local interconnect, poly, and oxide layers. Here, `Li1` and `Li2` are two local interconnect layers, `PO` is a poly layer, and OD is an oxide layer. Abstract Generator uses this information during extraction to determine where to extract next. Starting from a given shape, the extractor looks at the connectivity defined for the layer on which that shape is present to determine the layers to look for shapes on next. **Note:** Bridge vias are also supported during extraction.

© 2023 Cadence Design Systems, Inc.

Field	Description
Pin Geometry Restriction	This section provides you with finer control over final pin geometry.
Layer	Lets you type in the layer you want to assign for pin geometry. However, this layer must still be present in the technology file. If it is not, an invalid layer warning is displayed.
Geometry Specification	Lets you enter the geometry specification for the regions where pins are either to be restricted to or avoided.
Restriction	Lets you choose whether pins are to be allowed or avoided (*Allow* or *Avoid*) in the geometry specified in the *Geometry Specification* field. The specified geometry specification (G-spec) describes the region where pins are to be allowed or avoided.

Related Topics

Specifying Layers and Geometries for Extraction

Extracting Power Nets in Standalone Abstract Generator

Processing Antenna Information in Standalone Abstract Generator

Specifying Connectivity Information in Designs

© 2023 Cadence Design Systems, Inc.

Save Dynamic Abstract Generation Options Form

Use the Save Dynamic Abstract Generation Options form to specify the advanced abstraction rules.

The form contains the following tabs.

Tab	Description
General	Lets you specify the pin, PR boundary and blockage settings.
Extraction	Lets you specify the extraction information for signal and power nets
Antenna	Lets you create process antenna data for Pcells. By default, all the check boxes on this tab are deselected.

© 2023 Cadence Design Systems, Inc.

General

The following table describes the fields available on the *General* tab of the Save Dynamic Abstract Generation Options form.

Field	Description

Map Text Labels to Pins field	Specifies the text labels that you want to map from the layout view to be created as pins in the layout dual view. This field is case-sensitive. The syntax for specifying the text labels is as follows:
	`(textLPP1 geomLPP1 geomLPP2)`
	`(textLPP2 geomLPP3 geomLPP4)`
	where,
	■ The syntax of `textLPP` is `(textLayer textPurpose)` or `textLayer`.
	■ The syntax of `geomLPP` is `(geomLayer geomPurpose)` or `geomLayer`.
	An example of this is as follows:
	`(textLayer (metal1 drawing) (metal2 drawing))`
	In this example, the text labels on layer `textLayer` for all purposes are mapped first to the shapes on layer `metal1` with purpose `drawing`, and then to shapes on layer purpose pair `metal2 drawing`.
	Instead of specifying a layer purpose pair, if you specify only the layer name, 'all' purposes for the specified layer are considered during Dynamic Abstract Generation. See Text Labels-to-Pins Mapping.

Power Pin Names	Specifies a list of names for power pin. Specifying the pin names enables you to distinguish the pin types in the design. You can specify a list of pin names in the form of regular expressions, with each pin name separated by a space. See Regular Expressions for Creating Pin Names.
Ground Pin Names	Specifies a list of names for ground pin. Specifying the pin names enables you to distinguish the pin types in the design.
Clock Pin Names	Specifies a list of names for clock pin. Specifying the pin names enables you to distinguish the pin types in the design.
Analog Pin Names	Specifies a list of names for analog pin. Specifying the pin names enables you to distinguish the pin types in the design.

© 2023 Cadence Design Systems, Inc.

Field	Description
Output Pin Names	Specifies a list of names for output pin. Specifying the pin names enables you to distinguish the pin types in the design.
Geometry Search Depth	Specifies the hierarchy depth which is the number of levels in the design hierarchy the Dynamic Abstract Generation process must search for metal shapes that lie underneath a text label. The default value is `20`. You can control how far down the design hierarchy does Abstract Generator search for text labels during text-to-pin mapping.
Preserve text labels	Retains the text labels from the layout view in the layout dual view.

Using geometry on layers	Specifies a list of layers or layer purpose pairs to be used to define the prBoundary. The boundary is drawn so that it encloses all the geometries found on the specified layers or layer purpose pairs. The syntax for specifying the layers is as follows:
	`LPP1 LPP2`
	An example of this is as follows.
	`metal1 (metal2 drawing)`
	In this example, all geometries on layer `metal1` for all purposes and on layer `metal2` for `drawing` purpose are enclosed by prBoundary.

Blockage Spacing Constraint	Specifies a list of layer purpose pairs on which you want to define a spacing constraint along with the spacing value. For example, you can specify (`METAL1 0.75`) (`METAL2 0.3`). If you do not specify the value for the spacing constraint, than the default spacing value is the minSpacing value that is defined in the technology file for that particular layer. You can also specify the blockage spacing constraint in the Dynamic Abstract Generation options file as follows. `absSetOption( "AbstractBlockageUserDefinedSpacing" "(MET1 0.75) (MET2 0.3)")`

Extraction

The following table describes the fields available on the *Extraction* tab.

Field	Description
Signal Extraction	This section lets you specify the options for extracting signal nets.

© 2023 Cadence Design Systems, Inc.

Field	Description
Extract Signal Nets	Runs connectivity extraction on signal nets. By default, the *Extract Signal Net* check box is deselected.
Extract Layers	Specifies the layers or layer purpose pairs on which signal net extraction is to be performed.
Weak Layers	Specifies the layers that you want to be considered as weak layers. When extracting signal nets, you can set certain layers as weak layers and instruct the extractor to extract through these as well. The weakly connected pins will be created for the layers specified in the *Weak Layers* field. By default, the *Weak Layers* field is empty.
Pin Layers	Specifies the layers on which you want the pins to be created. This can be a subset of layers that are specified in the *Extract Layers* field.
Extraction Depth	Specifies how many levels down the instance hierarchy the Dynamic Abstract Generation process must search for the shapes to be extracted. The default value is `32`, which is the maximum number of levels in a design hierarchy stored in the database.
Extract Must Join Terms	Establishes a must-join relationship between all disjoint groups of geometries extracted from separate pins on the same net.
Must Join Term	Specifies a list of terminal names, each separated by a space. If the extractor finds a terminal name that matches one of the specified terminal names, and if there are disjoint geometries from different pins on the net after extraction, a `MUSTJOIN` relationship is created for that terminal.
Layer connectivity	Specifies the connectivity between layers. This enables you to specify the layer connectivity without having to add via definitions in the technology file. The default value of this field is based on the connectivity derived from analyzing the vias in the technology file. The format for specifying a value in this field is to list the layers as shown below. `(layer1 layer2 layer3) (layer4 layer5 layer6) ... (layer7 layer8) ...` In the example above, a triple layer entry means that `layer1` connects to **`layer2`** through the via layer `layer3`. A pair means that the first layer, `layer7`, connects to the second layer, `layer8`, by overlap (for example, local interconnect to polysilicon). See Restrictions for Signal and Power Pin Geometry.

© 2023 Cadence Design Systems, Inc.

Field	Description
Power Extraction	This section lets you specify the options for extracting power nets.
Extract Power Nets	Runs connectivity extraction for power nets. If this check box is selected, all the pin shapes created during the Pins step are extracted. By default, the *Extract Power Nets* check box is deselected.
Extract Layers	Specifies the layers or layer purpose pairs through which you want the power nets to be extracted.
Pin Layers	Specifies the layers on which you want the pins to be created. Creating pins on a layer helps to determine the shapes found by the extractor that are converted into pins.
Extraction Depth	Specifies how many levels down the instance hierarchy the Dynamic Abstract Generation process must search for the shapes to be extracted. The default value is `32`, which is the maximum number of levels in a design hierarchy stored in the database.

Antenna

The following table describes the fields available on the *Antenna* tab.

Field	Description
Hierarchical Antenna, Antenna for Input Pin, Antenna for Output Pin, Antenna for In-Out Pin, Extract Metal Area, Extract Metal Side Area	Specifies that Dynamic Abstract Generation computes the process antenna data for a given layer by using the following options on the *Antenna* tab. ■ *Hierarchical Antenna* ■ *Antenna for Input Pin* ■ *Antenna for Output Pin* ■ *Antenna for In-Out Pin* ■ *Extract Metal Area* ■ *Extract Metal Side Area*
Diffusion Layers	Extracts diffusion layers specified in the technology file.
Antenna Layers	Specifies the layers for which you want to calculate their process antenna data.

© 2023 Cadence Design Systems, Inc.

Field	Description

Define Gate Area, *Define Drain Area*	Specifies the layer-geometry specification for defining the gate and drain geometry regions, respectively. An example that defines the gate area region is as follows:
	`(poly and fiff)`
	An example that defines the drain region area is as follows:
	`(diff andnot poly)`

Define Oxide Number	Selects the oxide model to be extracted for a particular diffusion layer. The list contains four oxide types: `Oxide1`, `Oxide2`, `Oxide3`, and `Oxide4`. You can apply a unique oxide model for up to four diffusion layers.

Related Topics

Specifying Advanced Dynamic Abstraction Rules

© 2023 Cadence Design Systems, Inc.

Select Cells Form

Use the Select Cells form to select one or more cells using a variety of selection criteria.

Field	Description
Name	Lets you enter a regular expression that is included in the cell names that you selected. All matching cells are selected, irrespective of the *View* set.
View	Lets you select the view you are interested in. Only cells that have the specified *View* present and match any subsequent *Height*, *Property* or *Status* criteria are selected.
Height	Lets you specify minimum or maximum cell heights, or a range within which the cell height must fall.
Property	Lets you enter a system or user-defined cell property and a value for the cells you want to select.
Select By Error Messages	Lets you select one or more of the error messages listed in the table to select cells in a library. Only cells that have had a specific, selected error message applied to them will be filtered through in the selection procedure.
Name	Lets you enter a regular expression that is included in the cell names you want to select. All matching cells are selected, irrespective of the *View* set.

Related Topics

Selecting Cells Based on a Criteria in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Sort Cells Form

Use the Sort Cells form to sort the selected cells according to a variety of sorting criteria. The specified criteria are applied only to the selected cells displayed in the *Cell* pane and not to the overall library.

Field	Description
Method	Selects criterion for sorting from *None, Name, Height, Status,* or *Selection*. ■ *Name* sorts the selected cell list alphabetically by cell name. ■ *Height* sorts the selected cell list according to the heights of the cells in the specified *View*. ■ *Status* sorts the selected cells according to the status of the *View* specified, listing cells in a sequence: unknown, warning, error, and invalid. ■ *Selection* moves all the selected cells to the top of the cell table.
View	Lets you select a view for sorting. Select from *Layout*, *Logical*, *Pins*, *Extract*, or *Abstract*. The *Height* and *Status* methods consider only the view specified here. Cells that do not have the specified view are not sorted.
Order	Lets you select an order of sorting. Select from *Ascending* or *Descending*.

Related Topics

Sorting Cells in Abstract Generator

© 2023 Cadence Design Systems, Inc.

Specify SKILL Files Form

Use the Specify SKILL Files form to customize the Abstract Generator flow for specific tasks that cannot be performed using the options in the Abstract Generator form.

Field	Description
Add	Displays the Choose Skill Files form, which lets you select the required skill hook files.
Remove	Removes the selected SKILL hook files.
OK	Saves the selection and closes the Specify SKILL Files form.

Related Topics

Customizing Abstract Generator Flow Using SKILL Files

© 2023 Cadence Design Systems, Inc.

UnAbstract Pcells Form

Use the UnAbstract Pcells form to unabstract a Pcell in the layout window. Unabstracting a Pcell involves deleting the abstract data that exists for a Pcell.

Field	Description
All Pcells	Unabstracts all Pcells in the layout window.
Selected Pcells	Unabstracts the selected Pcells in the layout window.

Related Topics

Dynamic Abstract Generator Context-Sensitive Menu

© 2023 Cadence Design Systems, Inc.

B

Virtuoso Abstract Generator Environment Variables

This section provides information on the environment variables that can be used with the integrated Abstract Generator. The following environment variables are supported for public use.

List of Abstract Generator Environment Variables

- optionsFile
- showAbstractSection
- showAntennaOptions
- showExtractGspecTables
- showExtractSection
- showOverlapBoundaryOption
- showPinGroupingOptions
- showPinsSection

© 2023 Cadence Design Systems, Inc.

optionsFile

```
abstract optionsFile string "options_file_path"
```

Description

Specifies the path to the options file. Option files are a simple list of records in which each field occupies its own line and in which the records themselves are separated by blank lines.

The default value is an empty string (`""`).

GUI Equivalent

Command	*Tools – Abstract Generator – Design Information*
Field	*Options File*

Examples

```
envGetVal("abstract" "optionsFile")
envSetVal("abstract" "optionsFile" 'string "server1/abstract/Files/AG.options")
```

Related Topics

Specifying Design Information in the Abstract Generator

© 2023 Cadence Design Systems, Inc.

showAbstractSection

```
abstract showAbstractSection boolean { t | nil }
```

Description

Displays the *Abstract* section by default when you open the Abstract Generator form.

The default value is `t`. When set to `nil`, the section is hidden.

GUI Equivalent

None

Examples

```
envGetVal("abstract" "showAbstractSection")
envSetVal("abstract" "showAbstractSection" 'boolean nil)
```

Related Topics

Generating Abstracts

© 2023 Cadence Design Systems, Inc.

showAntennaOptions

```
abstract showAntennaOptions boolean { t | nil }
```

Description

Displays the *Gate* and *Drain* antenna options by default when you open the *Extract* section of the Abstract Generator.

The default value is `t`. When set to `nil`, the options are hidden.

GUI Equivalent

Command	*Tools – Abstract Generator – Extract* section
Field	*Gate* and *Drain*

Examples

```
envGetVal("abstract" "showAntennaOptions")
envSetVal("abstract" "showAntennaOptions" 'boolean nil)
```

Related Topics

Generating Abstracts

© 2023 Cadence Design Systems, Inc.

showExtractGspecTables

```
abstract showExtractGspecTables boolean { t | nil }
```

Description

Displays the options for extracting connectivity information for signal and power nets when you open the *Extract* section of the Abstract Generator form.

The default value is `t`. When set to `nil`, the section is hidden.

GUI Equivalent

Command	*Tools – Abstract Generator – Extract* section
Field	*Signal Layers*, *Pin Layers*, *Power Layers*, and *Use Signal GSpec For Power GSpec*

Examples

```
envGetVal("abstract" "showExtractGspecTables")
envSetVal("abstract" "showExtractGspecTables" 'boolean nil)
```

Related Topics

Generating Abstracts

© 2023 Cadence Design Systems, Inc.

showExtractSection

```
abstract showExtractSection boolean { t | nil }
```

Description

Displays the *Extract* section by default when you open the Abstract Generator form.

The default value is `t`. When set to `nil`, the section is hidden.

GUI Equivalent

None

Examples

```
envGetVal("abstract" "showExtractSection")
envSetVal("abstract" "showExtractSection" 'boolean t)
```

Related Topics

Generating Abstracts

© 2023 Cadence Design Systems, Inc.

showPinsSection

```
abstract showPinsSection boolean { t | nil }
```

Description

Displays the *Pins* section by default when you open the Abstract Generator form.

The default value is `t`. When set to `nil`, the section is hidden.

GUI Equivalent

None

Examples

```
envGetVal("abstract" "showPinsSection")
envSetVal("abstract" "showPinsSection" 'boolean nil)
```

Related Topics

Generating Abstracts

© 2023 Cadence Design Systems, Inc.

showOverlapBoundaryOption

```
abstract showOverlapBoundaryOption boolean { t | nil }
```

Description

Displays the *Create Overlap Boundary* option by default when you open the *Abstract* section of the Abstract Generator form.

The default value is `t`. When set to `nil`, the section is hidden.

GUI Equivalent

Command	*Tools – Abstract Generator – Abstract* section
Field	*Create Overlap Boundary*

Examples

```
envGetVal("abstract" "showOverlapBoundaryOption")
envSetVal("abstract" "showOverlapBoundaryOption" 'boolean nil)
```

Related Topics

Generating Abstracts

© 2023 Cadence Design Systems, Inc.

showPinGroupingOptions

```
abstract showPinGroupingOptions boolean { t | nil }
```

Description

Displays the options for grouping signal and power nets by default when you open the *Abstract* section of the Abstract Generator form.

The default value is `t`. When set to `nil`, the section is hidden.

GUI Equivalent

Command	*Tools – Abstract Generator – Abstract* section
Field	*Signal Grouping* and *Power Grouping*

Examples

```
envGetVal("abstract" "showPinGroupingOptions")
envSetVal("abstract" "showPinGroupingOptions" 'boolean nil)
```

Related Topics

Generating Abstracts

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

C

Abstract Generator Menu Commands

This topic describes the menu commands and the associated forms in the Abstract Generator. Abstract Generator does not check the validity or correctness of the entered text, such as terminal name or layer name, against any syntax rules.

Related Topics

Abstract Generation Flow

Bin Preferences for Abstract Generation

Cell Preferences for Abstract Generation

© 2023 Cadence Design Systems, Inc.

Standalone Abstract Generator Menu Bar

This topic describes the menu commands and the associated forms in the standalone Abstract Generator. Abstract Generator does not check the validity or correctness of the entered text, such as terminal name or layer name, against any syntax rules.

Abstract Generator contains *File*, *Bins*, *Cells* and *Flow* menu items.

The following table describes the types of options available in the menu bar.

Menu	Types of Options
File	Use the commands in the *File* menu to specify all setup information, ranging from defining technology information and specifying a library through importing physical and logical data for the cells and blocks to be processed, for Abstract Generator to process a library. The commands for specifying this information are also available on the toolbar. Other commands in the *File* menu let you export LEF descriptions of abstracts, generate a statistics report for the current library, and save the option settings of the current session. You can also replay a previous session.
Bins	Use the commands in the *Bins* menu to create, rename, and delete user bins, and to access the *Bin Options* form.
Cells	Use the commands in the *Cells* menu to select and sort the cells displayed in the cell table, move cells between different bins, view cell status reports, and set cell and terminal properties. The *Edit* submenu opens the Layout Editor, which you can use to inspect or modify any cellview.
Flow	Use the commands in the *Flow* menu to provide access to the main abstract generation flow commands: *Pins, Extract*, and *Abstract*. You can also access these commands by using the last four buttons on the toolbar. The flow options are available only when technology data has been successfully validated by the Abstract Generator technology checker and you have selected one or more library cells to be processed.

Related Topics

File Menu in Standalone Abstract Generator

Bin Preferences for Abstract Generation

Cell Preferences for Abstract Generation

© 2023 Cadence Design Systems, Inc.

Standalone Abstract Generator Interface

© 2023 Cadence Design Systems, Inc.

Dynamic Abstract Generator Context-Sensitive Menu

When the *Pcells* filter is selected from the *Show* drop-down list in the Navigator assistant, the *Abstract Status* column is displayed in the Navigator assistant. This column displays the dynamic abstract status for each Pcell by using an appropriate status icon.

The following table describes the dynamic abstract status icons.

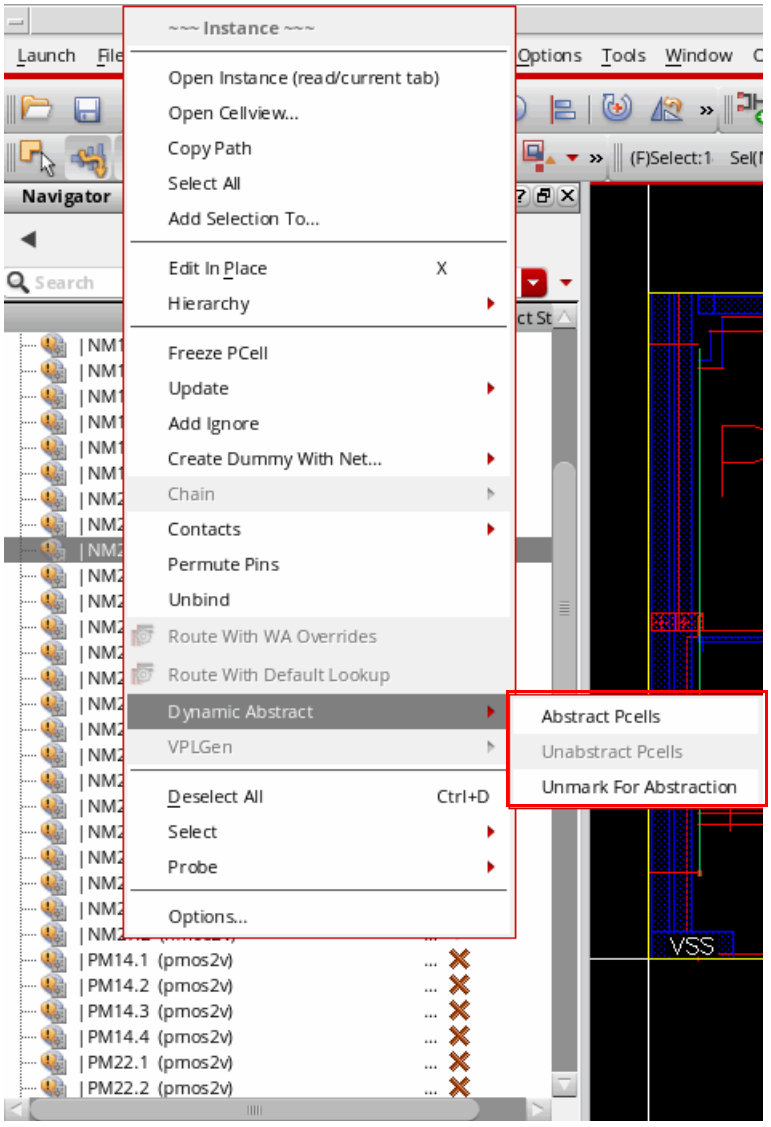
Icon	Name	Description
	Unmarked for abstraction	Indicates that the Pcell is unmarked for abstraction
	Dynamic Abstract does not exist (Marked for abstraction/ Unabstracted Pcell)	Indicates that the Pcell is marked for abstraction, but the abstraction data does not exist for the Pcell
	Dynamic Abstract exists (Abstracted Pcell)	Indicates that the abstraction data exists for the Pcell

You can right-click a Pcell in the Navigator Assistant and choose from the context-sensitive menu the required Dynamic Abstract Generation commands.

Depending on the abstract status of a Pcell, the context-sensitive menu provides the *Dynamic Abstract* commands for Dynamic Abstract Generation. For example, if a Pcell displays the abstract status as `Dynamic Abstract does not exist`, the context-

© 2023 Cadence Design Systems, Inc.

sensitive menu for the Pcell contains commands for to unmark the Pcell for abstraction, abstract the Pcell, and unabstract the Pcell, as shown in the figure below.

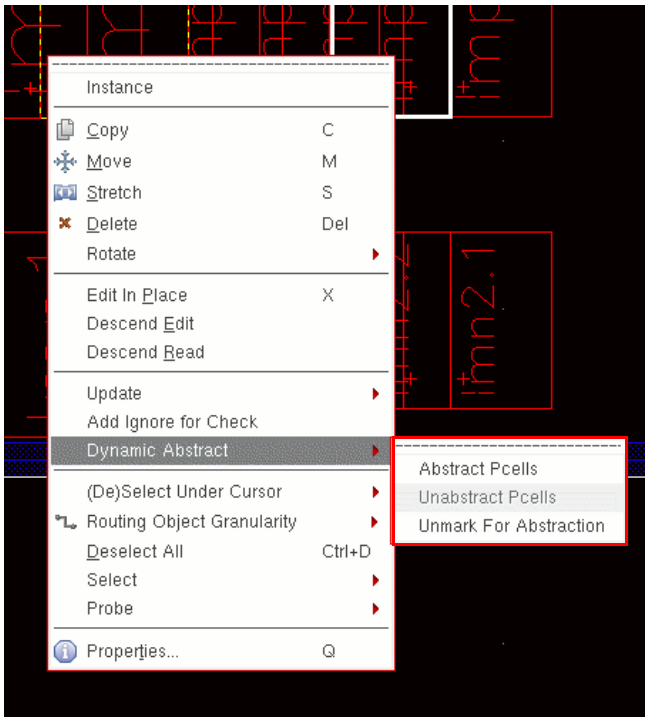


Selecting the *Abstract Pcells* command from the context-sensitive menu does not display the Abstract Pcells form. The abstract is created automatically based on the abstraction rules specified in the options file.

© 2023 Cadence Design Systems, Inc.

Layout Canvas Using the Context-Sensitive Menu

You need to select a Pcell in the layout canvas to view the context-sensitive menu. The Dynamic Abstract submenu that is available on the context-sensitive menu contains commands that can be used for Dynamic Abstract Generation. Right-click a selected Pcell to view this menu, as shown below.



Selecting the *Abstract Pcells* command from the context-sensitive menu does not display the Abstract Pcells form. The abstract is created automatically based on the abstraction rules specified in the default options file, namely, `.dynamicAbstract.options`.

Related Topics

Dynamic Abstract Generator Context-Sensitive Menu

© 2023 Cadence Design Systems, Inc.

D

Library Design Guidelines

This topic contains guidelines that helps you optimize your library to reduce die size and maximize the effectiveness of the place-and-route software. These guidelines help to understand about the models the software uses and gives you some general tips on the best ways to model the cells in your library.

A properly designed cell library is one of the most important factors in obtaining optimum results from Innovus™ Digital Implementation System (Innovus). A poorly designed library can lead to increased run times, undesirable placements, poor routing, and ultimately, less than ideal performance.

The library design guidelines are written with Innovus in mind, many also apply to any advanced place-and-route tool. Try to adhere to as many of these features as possible in your library design.

Before detailing the library development guidelines, it is important for the designer of the library to keep in mind that minimizing cell sizes does not necessarily minimize the total die size. In fact, some of the suggestions can cause an increase in the size of the individual cells, but will decrease the overall die size due to reductions in routing among the cells. This topic does not detail the procedures for creating or validating your library.

There are some general rules for all cell types:

- Every logical pin must have a physical description that contains at least one port. Each port must have at least one physical geometry.
- All objects belonging to the cell must be inside the cell boundary.
- In some technologies, power and ground pins extend to the edge or outside the boundary.
- The edges of pins and blockages must be at least one half the minimum spacing away from the cell boundary.
- Minimizing cell size does not necessarily minimize die size. It is better to increase the cell size slightly to optimize routing than to minimize cell size without keeping routing considerations in mind.

© 2023 Cadence Design Systems, Inc.

For an optimal design, follow these guidelines:

- Determine the routing grid for each layer.
- Try to keep pin and blockages on routing grids.
- Ensure that pins will be accessible after all connections are made.
- Keep cell size a multiple of the greatest common denominator of grids in each direction.
- Create a site for each cell height and multiple of grid minimum.
- Create place-and-route boundaries around cells.

Related Topics

Guidelines for Defining Blockages in a Cell Library

Guidelines for Setting Cell Porosity and High Density in a Cell Library

© 2023 Cadence Design Systems, Inc.

Guidelines for Designing a Cell Library

This topic provides suggestions for optimizing your design at the library level or inter-cell level, to ensure that the individual cells work together as well as possible.

Optimize Cell Widths

- Avoid creating standard cells of the same type that vary greatly in width.

  This results in unbalanced row lengths. In general, the width of the largest standard cells should be no more than five or six times the width of smallest standard cell of the same type in your library.

  If you need more complex cells, it is better to double or triple the height of the cells and keep a reasonable cell width. Innovus can optimize the area utilization by mixing single- and multiple-height cells in the core area without creating gaps around multiple-height cells.

- All cell widths should be a multiple of the pin grid.

  If a cell width is not a multiple of the pin grid, even though its pins are on the grid, the cell will cause the cells adjacent to it to be misaligned with respect to the grid. This makes having pins on grid useless because the misalignment of the other cells causes their pins to be off-grid, thereby using more horizontal or vertical tracks than needed.

  Making the cell width a multiple of the pin grid can increase the size of the cell, but will yield a reduction in the total area of the die.

- Placement should also be done on a placement snap grid so that the edges of the cells are also on the grid.

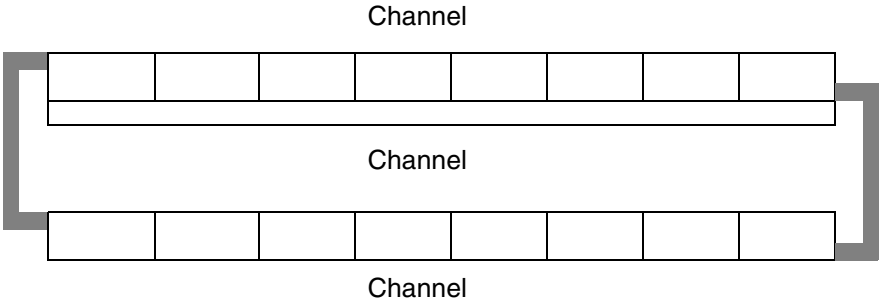
Optimize Cell Heights

Optimal cell heights vary depending on the row style your design uses.

Of the four styles described below, using the second style is highly discouraged. You can use any of the other three for Innovus. The last two row styles typically result in the smallest die sizes.

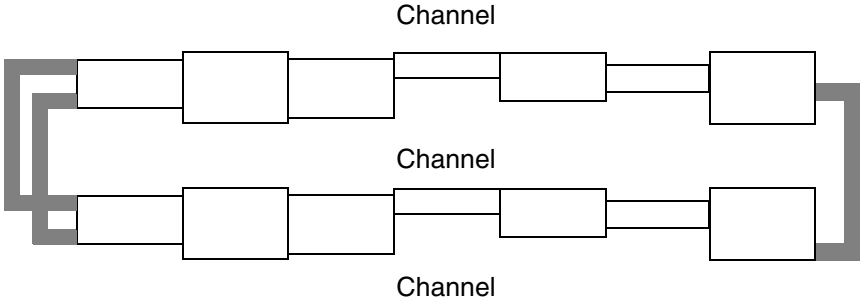
- **Style 1:** Standard height cells in single cell rows

© 2023 Cadence Design Systems, Inc.



Typically, these designs have two or more layers. Cell height is limited. You can have power sharing if you design it in. Power sharing can reduce die size by reducing the number of channels.

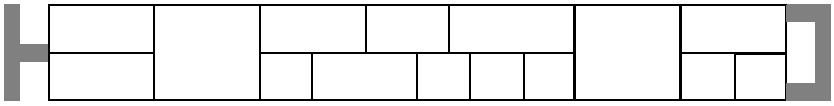
- **Style 2:** Variable height cells, in single cell rows



These designs are usually two-layer designs. They are not efficient for three or more layers. They allow cell heights to vary by about 30%, but do not allow for power sharing.

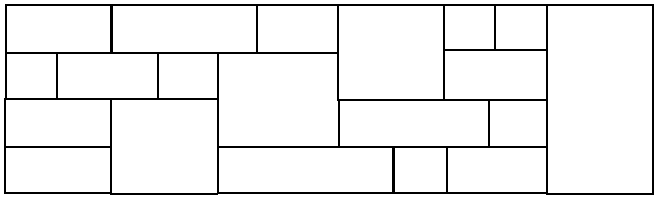
© 2023 Cadence Design Systems, Inc.

- **Style 3:** Single-height and double-height cells, in double cell rows



These designs have two or more layers. Single-height or double-height cells are arranged in double rows. This kind of design is meant for power sharing.

- **Style 4:** Multi-height cells in a sea-of-cells design



These designs feature three or more layers. They do not allow two-layer routing and have no channels, so you cannot use a channel-based router. You can have single-, double-, or multi-height cells. This kind of design permits the most efficient power sharing of both power lines.

Optimize Porosity

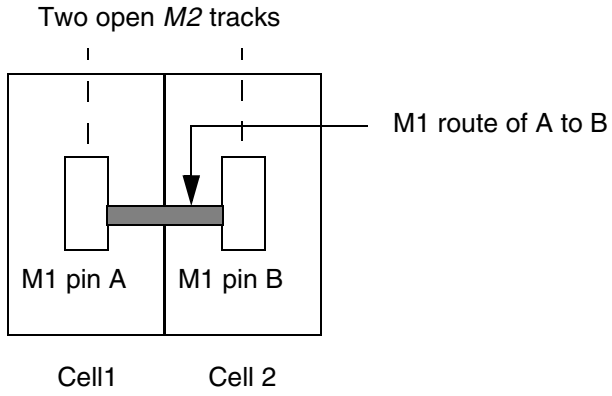
- Medium and large cells should have the same porosity, primarily in *M2*.

  This means that the percentage of free *M2* tracks should be the same for all the medium and large cells in the library. If the porosity is not uniform, the router might have trouble finishing if cells with low porosity are placed near each other.

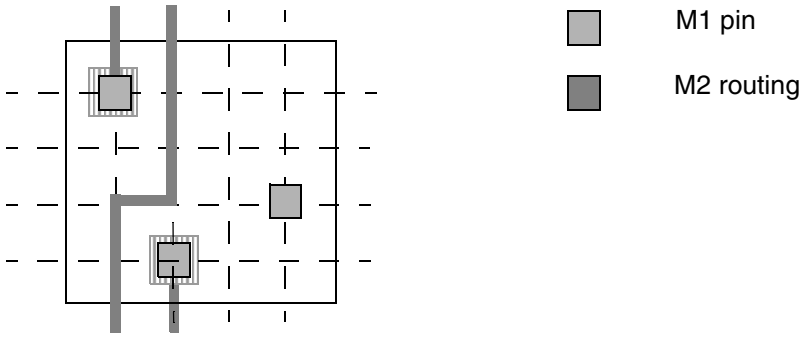
  For small cells (one to three tracks wide) you can let the placer optimize porosity by spacing them apart where necessary. However, careful design can improve routability significantly.

© 2023 Cadence Design Systems, Inc.

For example, you can maximize *M1* left/right access to pins on small cells, as shown in the following figure. This optimizes *M2* porosity by maximizing the amount of routing done on *M1*. Small cells often have this kind of routing.



You can also design small cells to have multiple accessible pins or open grids so that you can get Z-shape *M2* routes where possible, as shown in the following example.



Related Topics

Guidelines for Designing Pins

© 2023 Cadence Design Systems, Inc.

Guidelines for Designing Pins

This topic contains guidelines for designing pins so that you maximize the routability of the cells and improve the run time and memory performance of the router. The more porous the design, the easier it is for the router to make connections.

Guidelines to Optimize Pin Placement

Every pin should cover at least one grid intersection because the software connects most efficiently to pins at grid intersections. If you have off-grid pins, the software creates a pseudo-grid through the center of each off-grid pin.

Use the following guidelines to optimize pin placement:

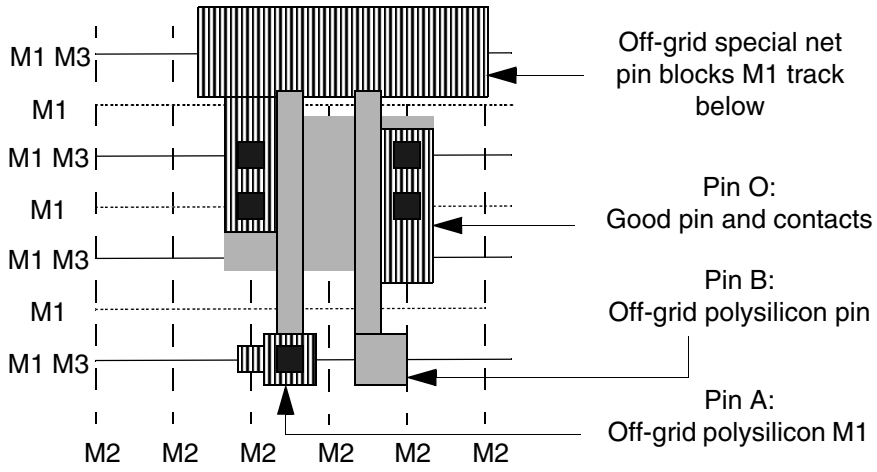
- Place pins on-grid wherever possible.
- Ensure that off-grid pins follow spacing rules by using the pseudogrid point at the center of the pin geometry.
- Center the pin via on the routing grid for its layer.
- Place special pins in such a way so that they do not obstruct routing on an adjacent track. You can place special pins off-grid for this.

Innovus has gridless routers but they handle gridded pins better than off-grid pins. A pin is considered to be on-grid if it meets the following criteria:

- The pin rectangle is centered at the grid intersection.
- The pin rectangle encloses the grid intersection point on all four sides by half the minimum metal width of that pin shape layer. The grid intersection is from the x and y routing grid pitch for that layer. The router routes to the pin rectangle only at that grid intersection.
- The router only routes to the center of the off-grid pin port rectangles if this is legal.

© 2023 Cadence Design Systems, Inc.

The following figure shows a LEF standard cell with shapes similar to layout data.

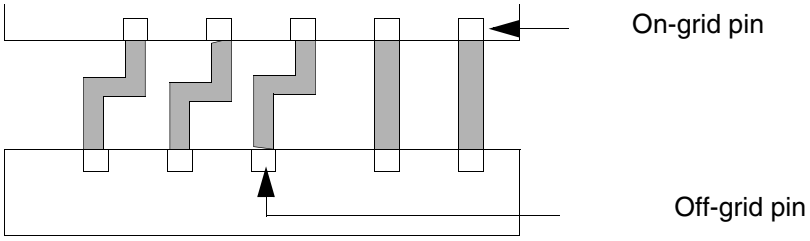


Pin A has several off-grid shapes and one on-grid shape. Pin B has two off-grid polysilicon pin shapes. Pin O has one on-grid shape. The off-grid shapes slow down the router and lead to inefficient routing. Further, if the spacing rules for this example did not allow *M1*/*M2* vias to be placed on pin A, the router would not be able to connect to Pin A. This could result in an inefficiently routed design.

Off-grid pins can affect your design in a number of ways. Some of these are listed below:

- Large die size

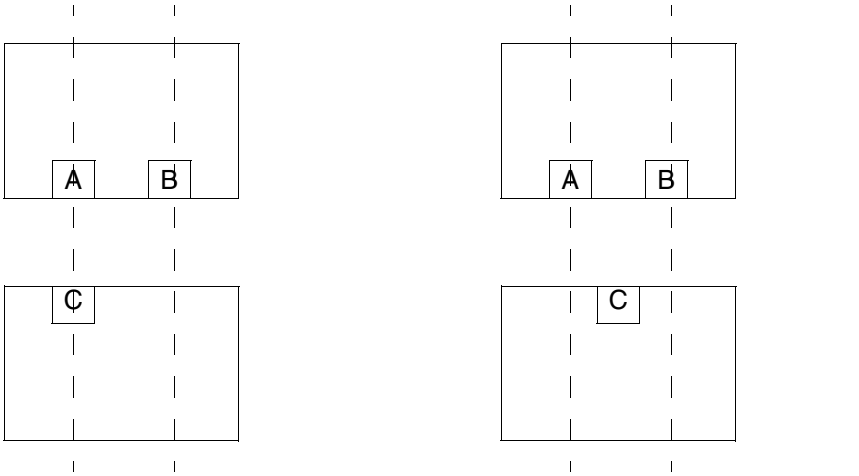
  The router creates jogs to compensate for pins that are not aligned. This forces the channels to expand vertically and increases the die size.



© 2023 Cadence Design Systems, Inc.

- Overconstrained design

  The more constraints there are, the harder it is for the router. The following figure shows how one off-grid pin increases the number of constraints.



On-grid pins: three pins, two constraints

A constrains C.
C constrains A.
B has no constraints.

Off-grid pin: three pins, four constraints

A constrains C.
C constrains A.
B constrains C.
C constrains B.

- Longer run-times

  This design slows down the router significantly because the router is more constrained. Each off-grid track can interfere with adjacent tracks.

- Optimistic global router results

  The global router might not recognize off-grid pins as blocking the routing track and, as a result gives you optimistic results.

Guidelines to Handle Special Pins

Special pins are pins routed by the special net router instead of the final router. Special pins have different shapes, depending on their function. On cells, you can have feedthrough or abutment special pins.

- Non-interference with adjacent grid lines.
- Define special pins in the `SPECIAL NETS` section of the DEF file.

© 2023 Cadence Design Systems, Inc.

- Place all power and ground feedthrough and abutment pins at the same axis on opposite sides of the cell.
- Position the power and ground feedthrough or abutment pins so that they align with corresponding pins in other cells.
- Avoid creating power and ground pins in *M2*, because *M2* within the cell makes it harder to route over the cell.
- Define a *shape* property for power and ground pins with the value set to either *feedthru* or *abutment*.
- Set the same y offset for power and ground pins from the cell origin so that the *FollowPins* command works properly. You can specify the origin in the LEF file or use the default, which is the lower left corner.

Guidelines to Maximize Routing Resources

The more routing resources you have available, the easier it is for the router to connect. You must maximize the resources in your most constrained layer. In three-layer designs where *M2* is vertical and *M1* and *M3* are horizontal, *M2* is typically the most constrained layer.

The following are some suggestions:

- Maximize pin accessibility.

  Make cell pins as accessible as they can be by declaring the complete geometry of the pin as a pin. This will allow the router to connect to the best location on the pin as it sees fit, rather than forcing it to connect to a particular point.

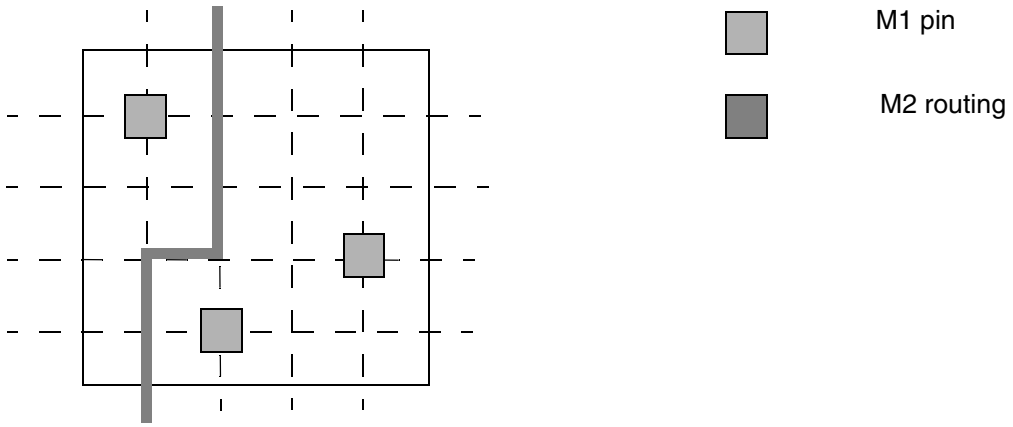
  Also, avoid having geometries on the same layer close to and around the pin, in effect, blocking it. This makes the router unable to make a connection to the pin.

  Special care should be taken not to block access to pins through vias. For example, avoid *M2* geometries too close to a *M1* pin such that *M1* to *M2* via cannot be dropped in order to access the pin on *M2*.

© 2023 Cadence Design Systems, Inc.

- Stagger pins in the x and y directions.

  Staggering pins provides additional vertical routing resources, as shown in the following figure:



  If two or more pins are aligned vertically, then the vertical track going through the pins can become blocked, forcing some of the horizontal routing resources to be used to connect to the pins.

  By staggering the pins in the vertical direction, especially at a pitch equal to integer multiples of the pitch of the vertical routing track, a higher degree of flexibility is given to the tool to connect to the pins in either direction.

- Minimize the number of geometries within the cell that are on the constrained layer.

  Any geometry on the constrained layer within the cell becomes a blockage to routing in that layer. For three-layer designs where *M2* is the vertical layer, you can maximize routing resources by creating internal cell geometries in *M1* and polysilicon, and leave *M2* and *M3* for routing.

- Allow extra routing resources in the constrained layer.

  In libraries for gate isolation base arrays that might be congested in *M2*, you can add extra *M2* routing resources as padding. For example, a two-input NAND gate might be three tracks wide with every track obstructed by a pin. You can model this as at least four tracks wide to provide an extra track for routing.

- Do not add pieces of metal to provide extra connection points.

  In some libraries, you add a piece of *M2* to a pin to provide the router with an extra connection point if *M1* is blocked. This is counter productive because unused *M2* areas become blockages that limit the flexibility of the router. You do not have to add extra metal, because the router can drop vias to the pins.

© 2023 Cadence Design Systems, Inc.

Related Topics

Guidelines for Designing a Cell Library

Design IO, Corner Cells, and Blocks

© 2023 Cadence Design Systems, Inc.

Design IO, Corner Cells, and Blocks

Abstract Generator uses the cover blockage model for IO cells, preventing the router from routing over IO cells.

Abstract Generator uses the same model for generating abstracts for corner cells.

- Maximize porosity for over-the-block routing.

  Leave unobstructed areas for over-the-block routing. Minimize the number of M2 and M3 geometries inside the block, so that these routing resources can be used to route over the block.

- Create power rings around large macro blocks.

  It might be easier to accommodate power routing around a macro block if you create a power ring around it and connect the internal buses to the ring.

- Pad around block boundaries.

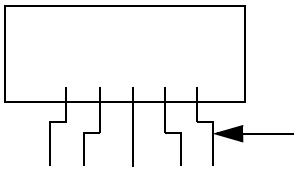
  It is always good to have extra routing resources around a block, because this area can get congested. Leave extra room around the corners.

- Create as much open vertical *M2* space as possible on the left and right sides.

- Create as much open *M1* and *M3* space as possible on the top and bottom.

- Space pins apart by at least line-to-via spacing.

  If your design requires line-to-line spacing, add flare outs to line-to-via spacing even if this sticks out from the macro, as shown in the following figure.



Related Topics

Guidelines for Designing a Cell Library

© 2023 Cadence Design Systems, Inc.

Guidelines for Setting Pitch in a Cell Library

This topic discusses the various guidelines associated with determining routing grid pitches and recommended pitch setting strategies.

You must define horizontal and vertical grids for routing on every routing layer. To do this, you must do the following:

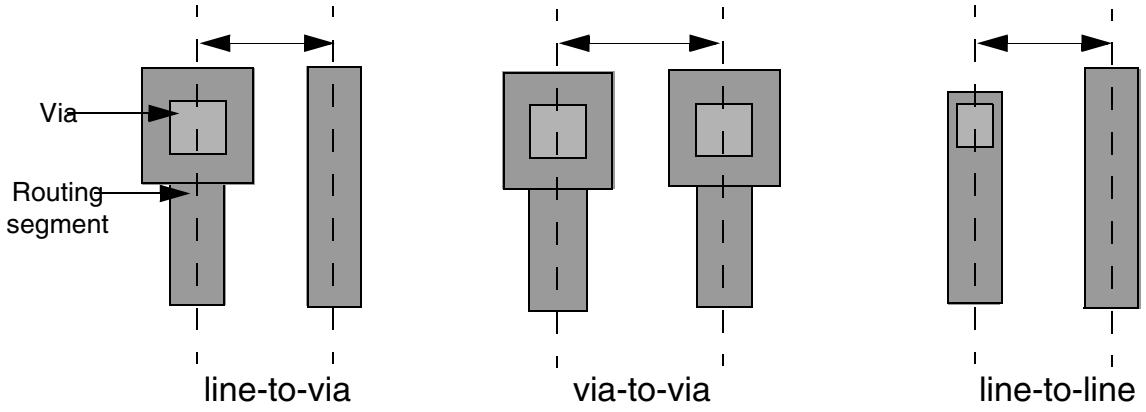
- Select the preferred and non preferred routing direction for each layer.

  One common convention is *HVH*, where the preferred direction for routing *M1* and *M3* is horizontal and the preferred direction for *M2* is vertical.

- Define a minimum pitch for each layer in the preferred direction track for that layer.

  This pitch must be at least line-to-via distance. Anything less will result in low utilization rates and longer run times, because the router will not be able to place vias next to wires on adjacent tracks.

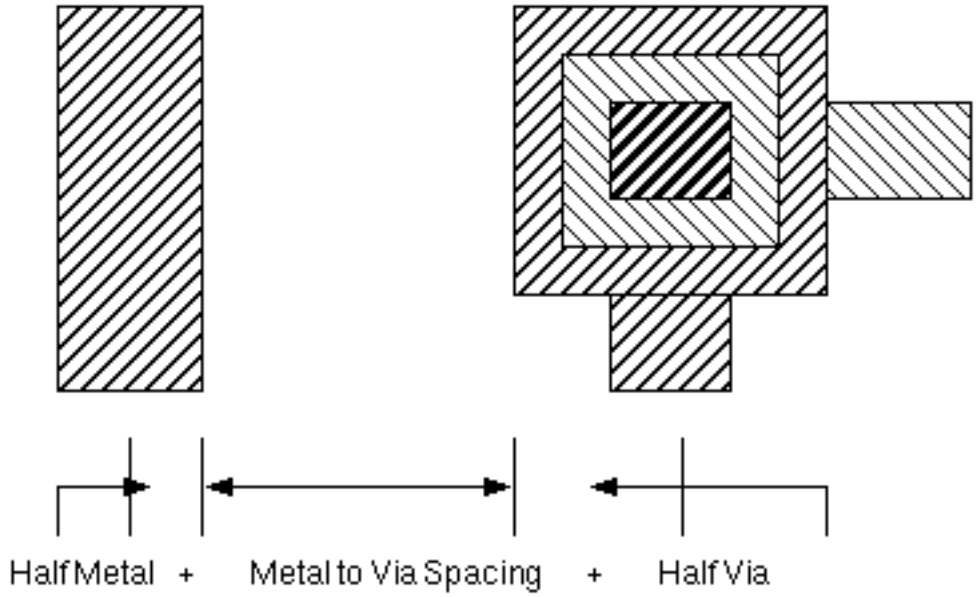
  The following figure shows line and via definitions for track pitch:



© 2023 Cadence Design Systems, Inc.

The most common track pitch is line-to-via, because it is the minimum pitch in which you can place a via next to a route. It is calculated as follows:

half the metal width for the layer + the metal spacing + half the via.



- Assign tracks in the non-preferred direction for every routing layer.

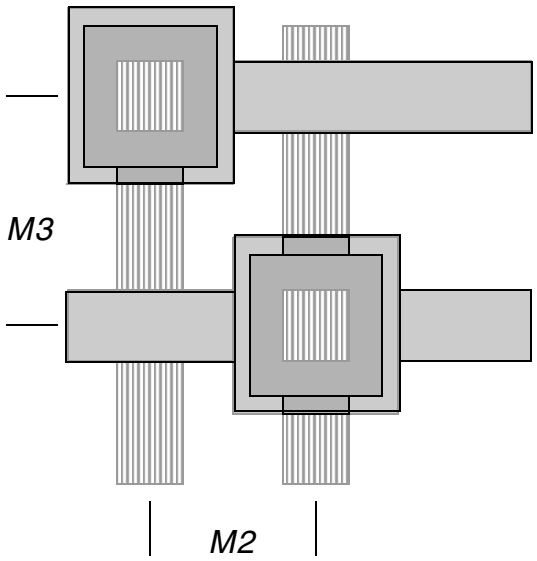
  Align these tracks with preferred direction tracks on adjacent layers, even if this is different from the track pitch of that layer in the preferred direction. Lining up the tracks lets the software create jogs at locations where vias can be placed or where pins can be connected to consistent xy locations.

- Allow for diagonally adjacent vias.

  Increase spacing to allow vias to be placed diagonally adjacent to each other. This might make the orthogonal spacing slightly more than line-to-via, but will reduce overall die

© 2023 Cadence Design Systems, Inc.

size, because lack of via resources causes more congestion and longer routes to route around congested areas.



- Maximize total routing track resources

  If you have an HVH layer scheme, keep *M1* and *M3* spacing to the minimum, and make any needed spacing increases in M2. Minimizing spacing on two layers (*M1* and *M3*) gives you more total routing resources than minimizing spacing on one layer by the same amount.

  To a certain extent, the placer also adjusts for different horizontal and vertical demands on the routing resources. You can also use a rectangular die instead of a square one to maximize routing resources.

- Use rectangular vias if that allows you to have smaller track pitch and more total routing resources overall.

  Ensure that the rectangular via and pitch still allows diagonal vias.

  Normally, you maximize your total routing track resources. However, if one direction is constrained, you might want to maximize your resources in that direction.

  For example, take an HVH library with the same line-to-via spacing for all three layers and no diagonal vias. To allow for diagonal vias, increase the spacing for the horizontal layers (*M1* and *M3*) to maximize the vertical routing resources on *M2*. This might not optimize the overall routing resources, but it reduces the congestion on the critically constrained *M2* layer.

© 2023 Cadence Design Systems, Inc.

Alternatively, if the cells have no free *M1* tracks and the *M3* pitch is much larger than *M2*, you can increase the spacing on *M2*.

Another technique for handling an over constrained layer is to use a die aspect ratio that minimizes the routing demand for that layer.

Related Topics

Guidelines for Setting Pitch for Layers with the Same Preferred Direction

Guidelines for Setting Pitch for Via Stacking

Guidelines for Designing a Gridded Library for Innovus

Guidelines for Converting an Existing Library into Gridded Library

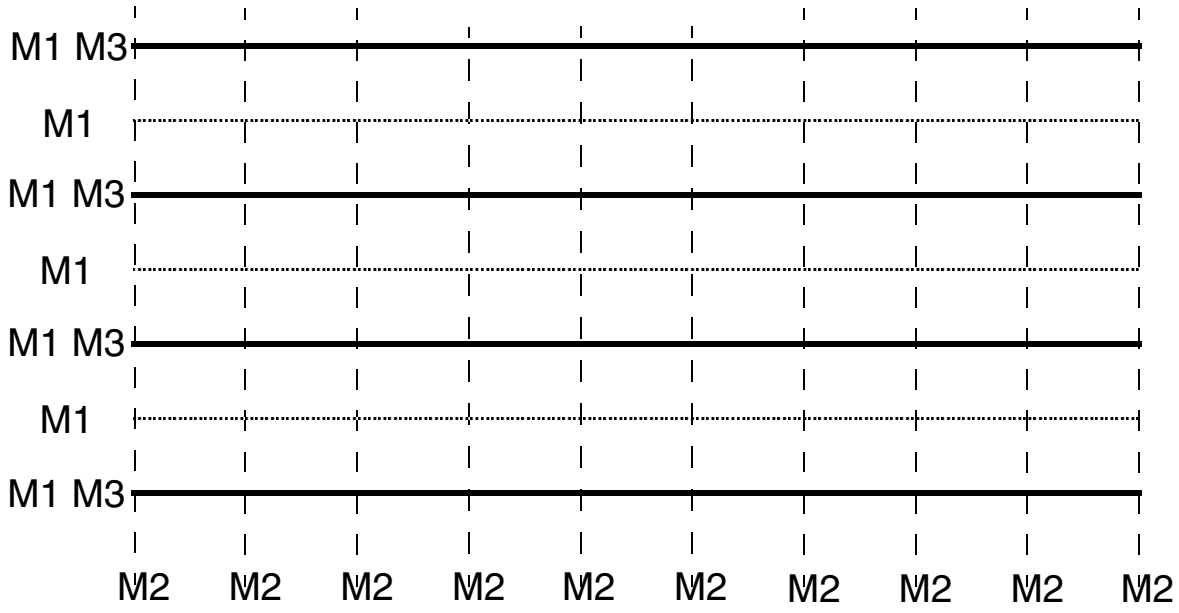
© 2023 Cadence Design Systems, Inc.

Guidelines for Setting Pitch for Layers with the Same Preferred Direction

Cells are more porous if there is free routing resource to route over the cell. A porous library allows high core utilizations. There are following guidelines for setting pitch for layers with the same preferred direction:

- Align the tracks on the layers as much as possible
- If you cannot align the tracks, keep the pitch ratio between them simple, for example, 1:2 or 2:3.

  The following figure shows a ratio of 2:1 between *M1* and *M3*. The figure shows preferred direction tracks. Non-preferred direction tracks should be the same.

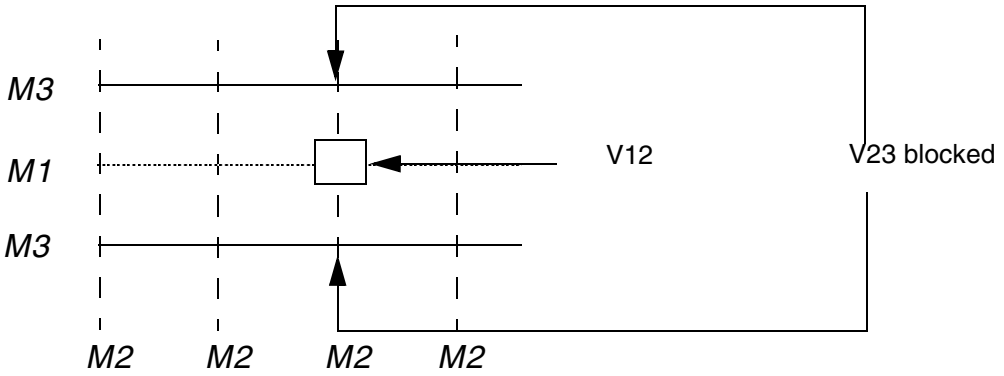


If *M1* and *M3* have complex ratios, such as 11:8, you can run into the following problems:

- If you have inter-layer spacing rules between cut layers, vias will often be limited on two tracks instead of one.

© 2023 Cadence Design Systems, Inc.

In the following figure, *V12* prevents the router from dropping *V23* vias above and below it on the *M3* tracks. If the *M1* and *M3* tracks were aligned, the via (*V12*) would block only one *V23* via.



- The above design can seriously affect memory allocation at run time. With a ratio of 11:8, you must create a non preferred *M2* track for every *M1* and *M3* track. Because these tracks align only at every eighth track, the result is more *M2* tracks than required with a 1:1 ratio, and this affects memory allocation.
- With a ratio such as11:8, your choice of cell heights is limited. This is because the cell height should be a multiple of the common denominator of the track spacing.
- If your technology allows center-stacked vias, a ratio like 11:8 will produce few.

Related Topics

Guidelines for Setting Pitch in a Cell Library

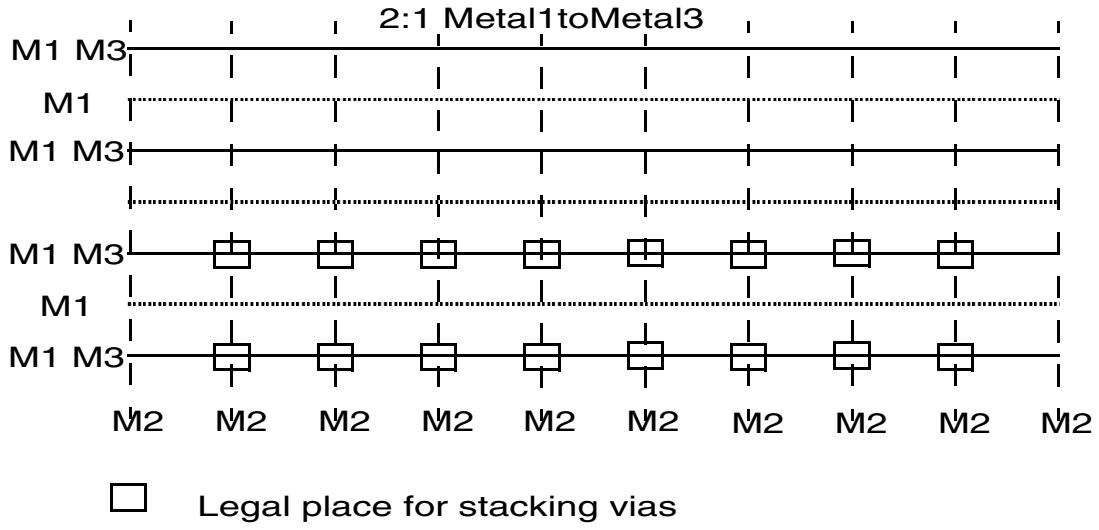
© 2023 Cadence Design Systems, Inc.

Guidelines for Setting Pitch for Via Stacking

If two metal layers have the same routing direction, make the relationship one of the following: 1:1, 1:2, 2:3, or 3:4.

Vias cannot be stacked where *M1* and *M3* are not coincident. You can only stack vias at grid intersection points where all layers in the stack exist. If the relationship is not one of the above, it is difficult or impossible to place vias.

If your process allows stacking of vias, you *must* use one of the above ratios. To stack vias everywhere, use 1:1. This is described in the following figure.



Related Topics

Guidelines for Setting Pitch in a Cell Library

© 2023 Cadence Design Systems, Inc.

Guidelines for Designing a Gridded Library for Innovus

Use the following general guidelines to simplify the internal data structures that Innovus uses during routing (assuming horizontal rows):

- Make cell width a multiple of the vertical routing track pitch.

  Cell widths should be a multiple of the vertical routing pitch if there is only one vertical routing layer.

- Make cell heights a multiple of the horizontal routing layer pitch.

  If there is more than one horizontal routing layer, use a multiple of the greatest common denominator of the horizontal track spacing.

- Create pins that cover routing grid intersection points by half the minimum width.

  The CPU time increase for on-grid versus off-grid routing varies from 20% to 30%, depending on how many off-grid pins are in the library. The quality of route is not significantly degraded with off-grid routing.

- Make blockages cover grid intersection points.

  Blockages and pins should cover grid intersection points even after the blockages are fractured.

  The routers can use any grid intersection point not covered by a pin or blockage shape.

Related Topics

Guidelines for Setting Pitch in a Cell Library

© 2023 Cadence Design Systems, Inc.

Guidelines for Converting an Existing Library into Gridded Library

The following guidelines apply if you have to create a routing grid for a library that was not designed with a routing grid in mind, or for a library that has a routing grid designed for an older technology that needs to be reduced.

- Calculate all minimum line-to-via and via-to-via pitches for all the layers. Calculate minimum xy distances for all diagonal vias.
- Determine the routing pitch and offset for each layer by examining some cell layouts from the existing library.

  You can get the *M2* preferred pitch (x coordinate) from the pin center-to-center distance of layout shapes. Getting the *M1* pitch (y coordinate) might be harder because many cells in older libraries were not designed with a y pitch in mind.
- If the designed x or y pitch is larger than the line-to-via minimum, reduce the pitch.

  It might be easier to reduce the y pitch than the x pitch. Reset cell boundaries to be a multiple of the new pitch.
- Define different *M1* and *M3* pitches to maximize routing resources.

  Remember to use simple ratios. The cell boundary must be a multiple of both layers.
- Use rectangular vias for routing if that allows the grid you want.

Related Topics

Guidelines for Setting Pitch in a Cell Library

© 2023 Cadence Design Systems, Inc.

Guidelines for Modeling Pins in a Cell Library

Consider the following guidelines while deciding pin layers:

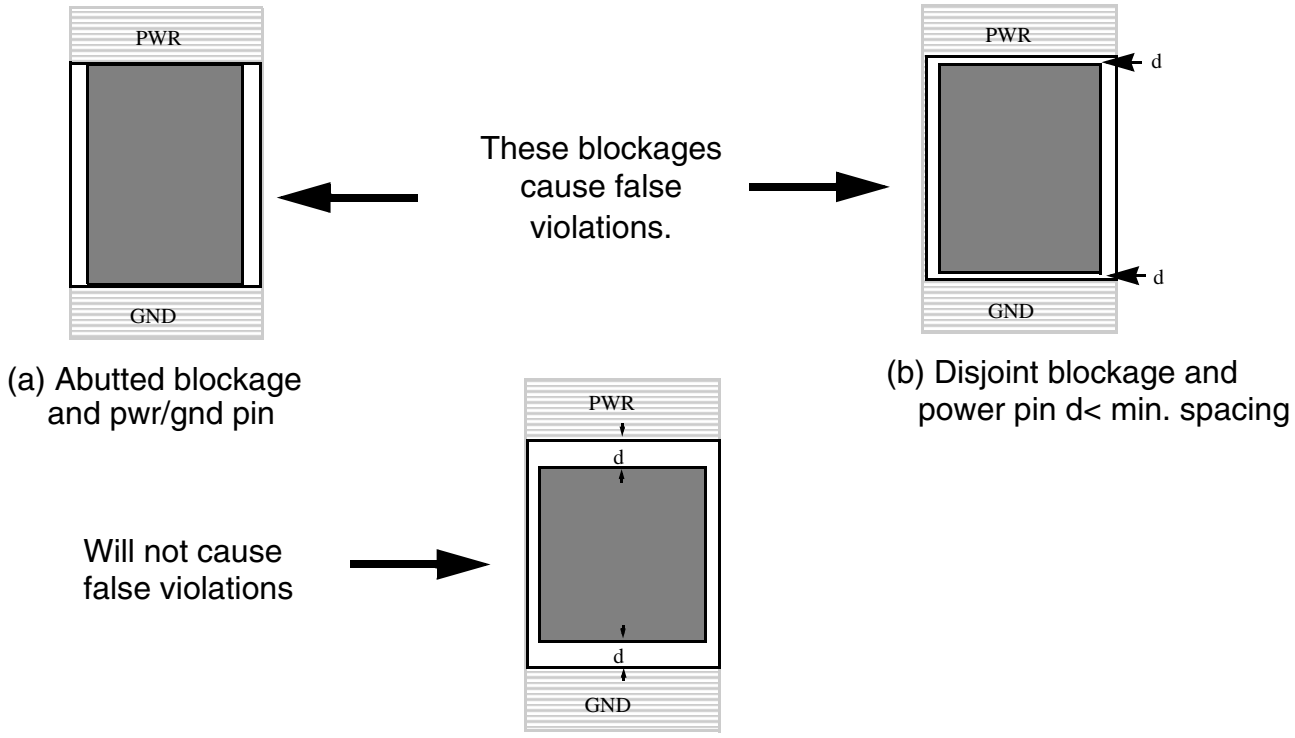
- Avoid creating *M2* geometry on pins. This blocks routing for other nets if the *M2* geometry is not required.
- Create an *M2* pin only if the pin is accessible only by *M3*. That is, it is blocked completely with *M1* and *M2*.

© 2023 Cadence Design Systems, Inc.

Power to Blockage Violations in Innovus

A common library problem occurs during the power and ground connections by the special router used c.

If the blockage does not keep a legal clearance to the power and ground pins, as shown in Figures (a) and (b) below, false violations are highlighted that make it difficult to identify real violations.

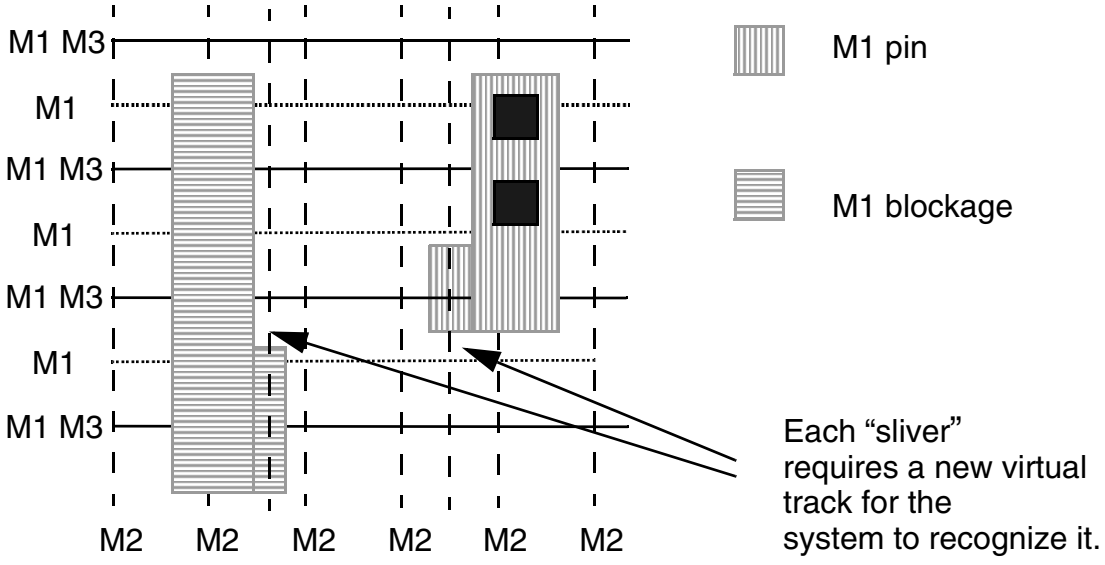


© 2023 Cadence Design Systems, Inc.

Virtual Tracks in Innovus

Each off-grid rectangle must be made on-grid in order for Innovus to recognize it. This is accomplished using virtual tracks.

Certain libraries require the creation of thousands of virtual tracks because of the presence of off-grid geometries, which must be avoided during routing. The following figure shows virtual tracks in use.



Off-grid routing on the virtual track might cause violations with fractured pin or blockage rectangles.

When virtual tracks are present, the router will use either virtual tracks or regular tracks, depending on which are closest.

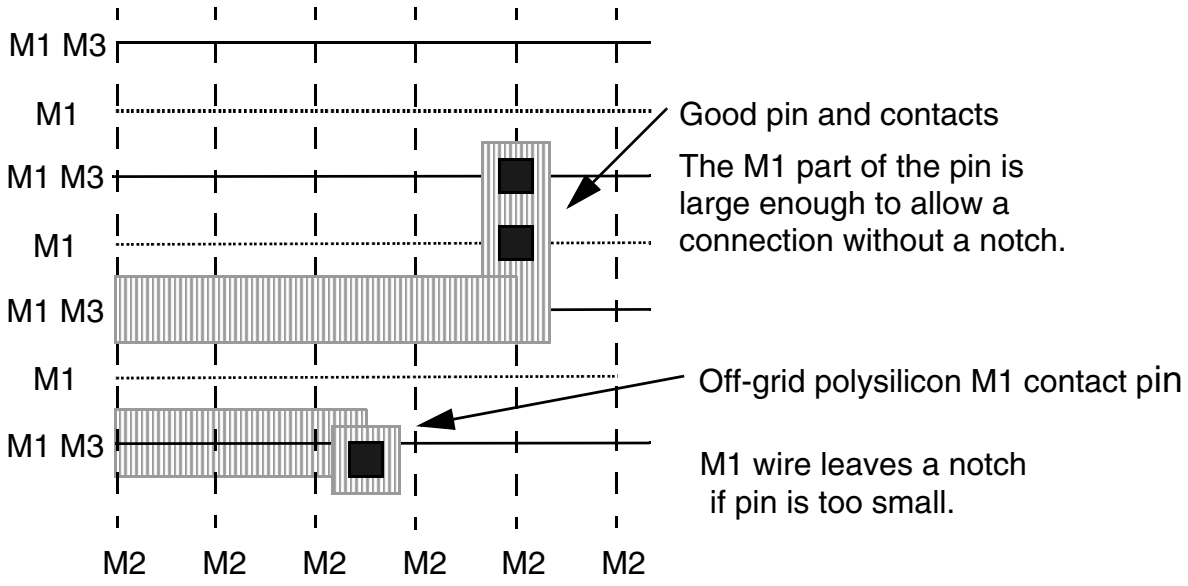
© 2023 Cadence Design Systems, Inc.

Guidelines for Centering On-grid Pins in Innovus

If off-grid pin access is not allowed, you should center pins on grid intersection points. If a pin shape touches a grid intersection point, it is considered on-grid.

If your pins touch grid points, and you do not allow off-grid routing;

- The router will recognize it as needing to be routed, and will avoid it when routing other nets.
- Depending on the pin shape and position, the router might be unable to route to the pin, or make an efficient connection.



Routing problems can be caused by incorrectly defined on-grid pins. These problems are sometimes referred to as off-grid problems. The router and verify geometry (in Innovus) do not consider a notch a violation. These violations are found by the `samenet` check.

© 2023 Cadence Design Systems, Inc.

Routing Issues in Pins

- The pin geometry interferes with the adjacent grid without covering it.

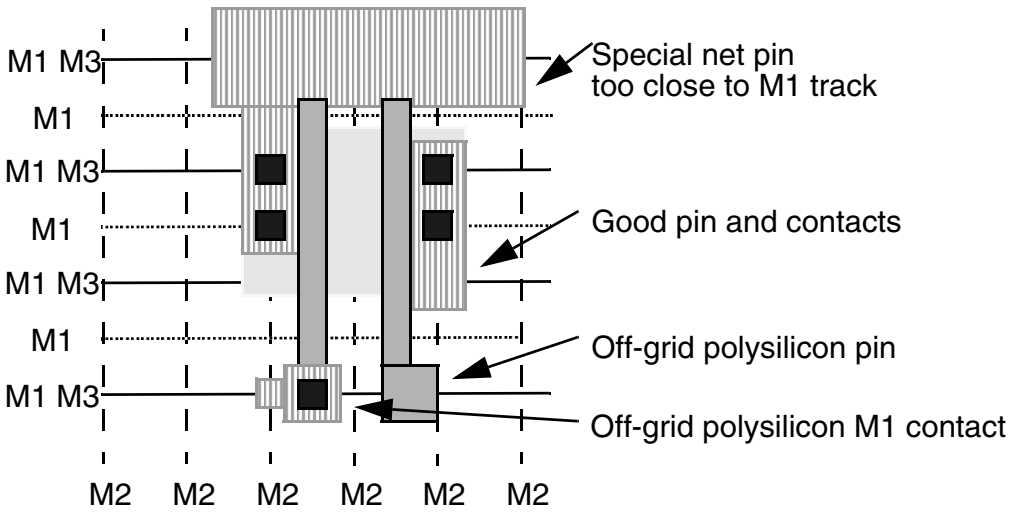
  The problem with off-grid pins that do not cover grid points but are too close to allow routing, is that the global router will not see the track as obstructed. The global routing scores will not consider that the track is blocked by the off-grid pin and will give you over-optimistic results.

- Vias (in pins) are not centered on the routing grids for its layers.

  If off-grid pins exist, and the router has been set to allow off-grid pins, then grids are created by the router. Any pin that does not cover the routing grid with enough area to place a via is considered off-grid, and a track is created through the center of the pins. Off-grid tracks will slow the router by requiring more memory. Routing is slower because wires on each off-grid track interfere with adjacent on-grid routing.

- Special pins need not be on-grid, but should follow blockage rules.

  The following figure shows potential routing problems.



Related Topics

Guidelines for Modeling Pins in a Cell Library

© 2023 Cadence Design Systems, Inc.

Guidelines for Defining Blockages in a Cell Library

All geometries on routing and cut layers that are not associated with a pin are identified as blockages by Abstract Generator. A blockage is a keep-out area for any routing layer or cut layer.

You can use Abstract Generator to create blockages. You can specify the *Geometry Specification* for the layers on which the blockage is to be created. Alternatively, from Abstract Generator you can open any one of the layout, pin, extract, or abstract cellview in VLS L and edit them to add blockages.

Use the following guidelines when blocking a routing layer:

- If all of the routing is blocked for a layer, then create a single blockage over the area instead of many separate blockages.

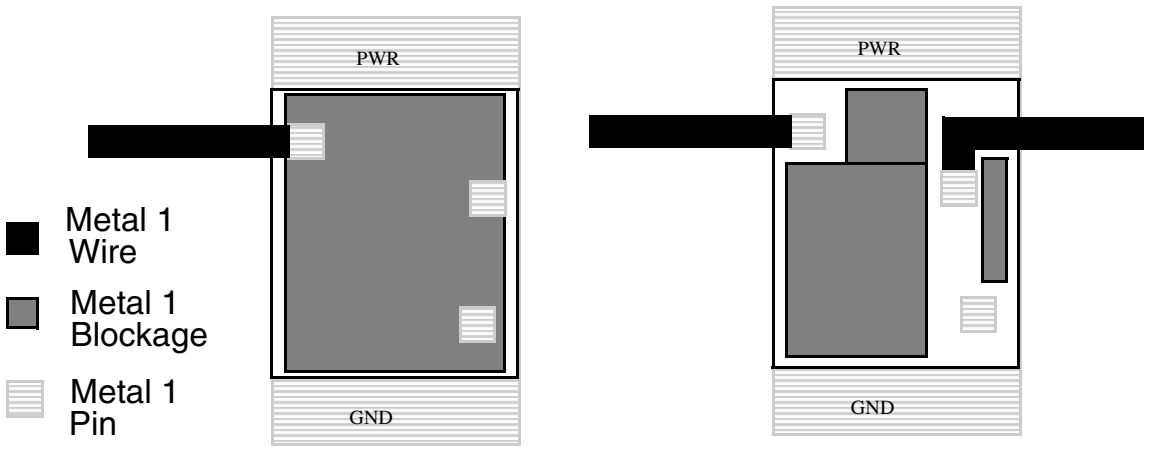
  This simplifies the LEF file generated for the abstract.

- When drawing a blockage over the whole cell, leave half a minimum spacing between the blockage and the boundary.

- If there are free tracks available, leave them open in the abstract.

  This can increase wiring density.

  In the following figure, the blockages in the cell on the right allow *M1* routing to the pins.



Full Metal 1 Blockage Partial Metal 1 Blockage

© 2023 Cadence Design Systems, Inc.

Related Topics

Creating Blockages

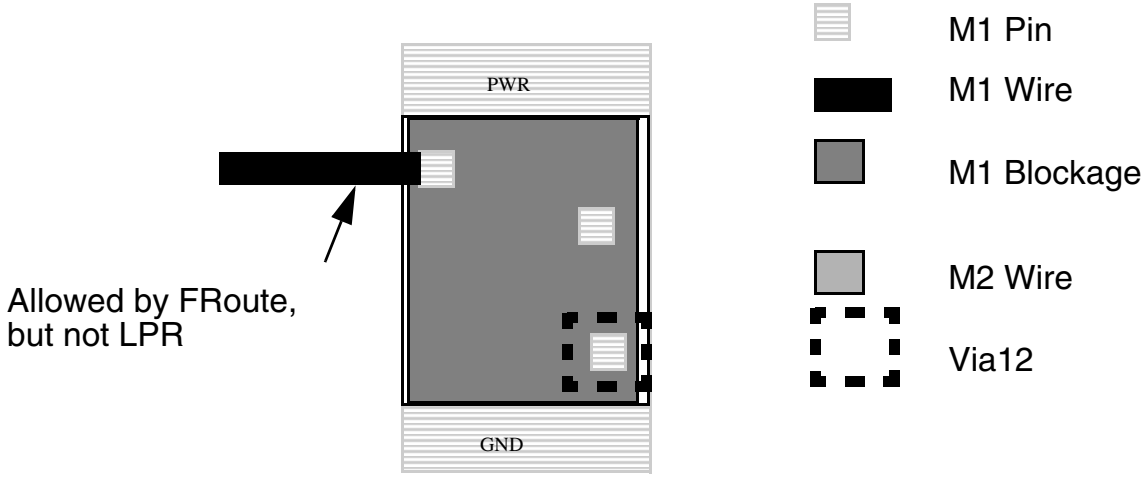
© 2023 Cadence Design Systems, Inc.

Pin Accessibility of Blockages in Abstract Generator

Pins are accessible from the side if they are less than one routing grid distance from the blockage edge, and the command `FROUTE.ALLOW.PINACCESS.ATBLOCKAGEEDGE` is set to `TRUE`. The command `FROUTE.ALLOW.PINACCESS.ATBLOCKAGEEDGE` controls *FRoute* only, and not the line probe router.

The line probe router (LPR) will *not* route to any pin that is surrounded by a blockage on that layer.

In the following figure, pins are accessible by routing over the cell with *M2*, and dropping a via to *M1* (Via12 is a via from *M1* to *M2*).



If your library has blockages defined as shown above, you must route the clock tree with *GRoute* and *FRoute*, and not *CRoute* Balance.

Related Topics

Guidelines for Defining Blockages in a Cell Library

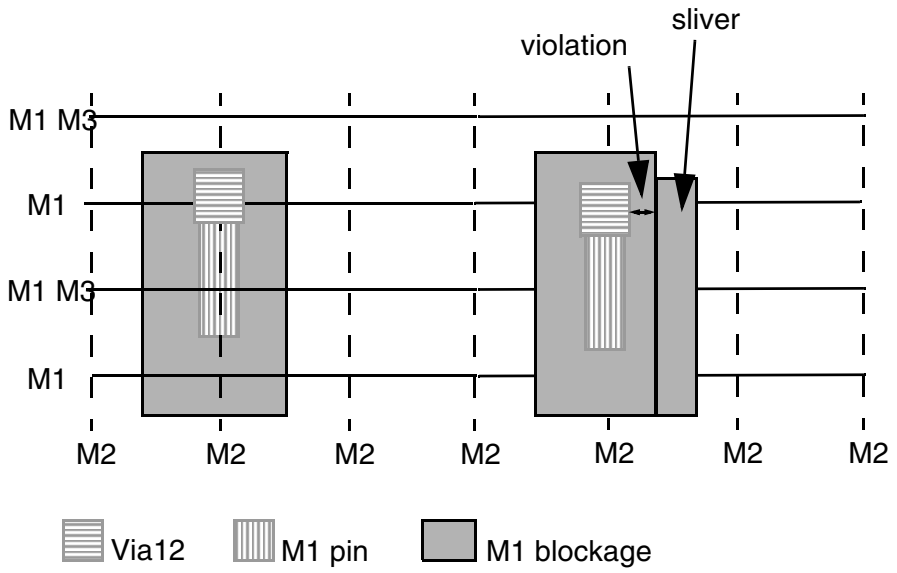
On-grid Versus Off-Grid Blockages in Abstract Generator

© 2023 Cadence Design Systems, Inc.

On-grid Versus Off-Grid Blockages in Abstract Generator

On-grid blockages completely cover the grid points that are interfered with.

Slivers that result from fracturing can cause edge-to-edge spacing violations. This happens when a sliver is off-grid, as shown on the right of the figure below. The sliver covers no grid intersection points, but interferes with routing.



The blockage on the left covers three grid points and effects only one vertical routing track. This is an on-grid blockage. It allows a via to be placed without violations. The blockages on the right cover three grid points and interfere with two vertical routing tracks. The sliver also causes an edge violation with the via. The sliver is an off-grid blockage.

Libraries with no off-grid blockages allow the best routing density. The final router runs faster if all blockages within cells are on-grid.

Related Topics

Guidelines for Defining Blockages in a Cell Library

© 2023 Cadence Design Systems, Inc.

Guidelines for Setting Cell Porosity and High Density in a Cell Library

Cells are more porous if there is free routing resource to route over the cell. A porous library allows high core utilizations.

There are following guidelines for setting cell porosity and high density in a cell library:

- Pitch Routing in Abstract Generator
- Pin Spacing in Abstract Generator
- Pin Staggering in Abstract Generator
- Tracks for Wide Wire Designs
- Variable-Height and Multiple-Height Cells
- Non-rectangular Blocks

Related Topics

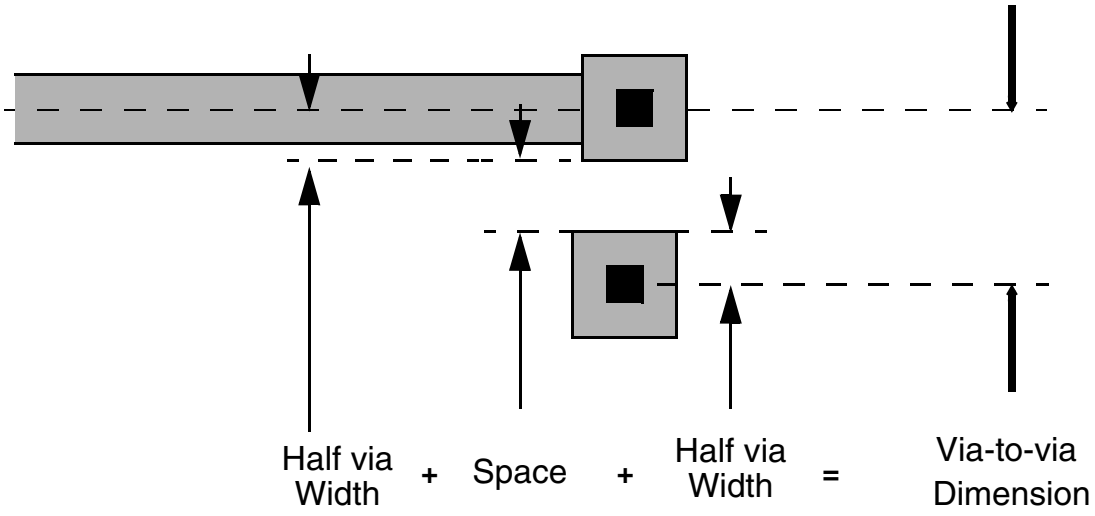
Guidelines for Setting Pitch for Layers with the Same Preferred Direction

© 2023 Cadence Design Systems, Inc.

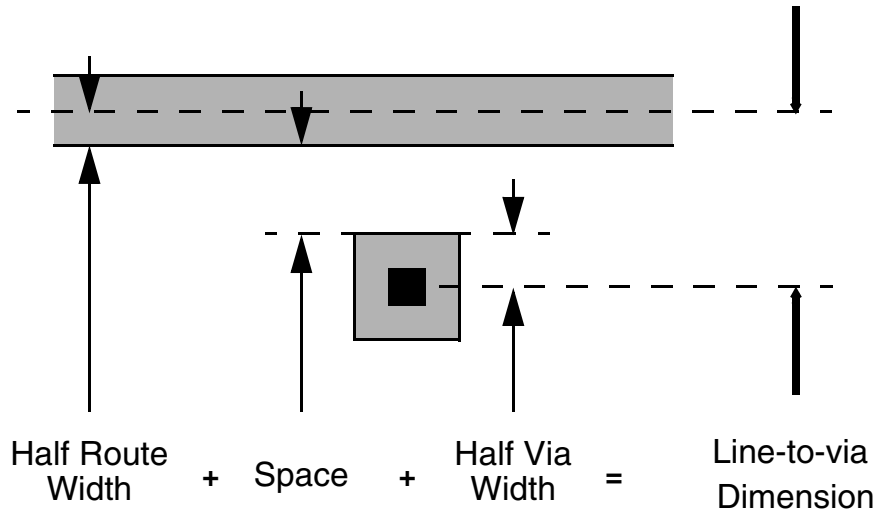
Pitch Routing in Abstract Generator

The pitch you choose affects the routing density. The pitch is usually a value between via-to-via spacing and line-to-via spacing.

Via-to-Via Spacing: This is the spacing required for adjacent vias and is illustrated in the figure below.



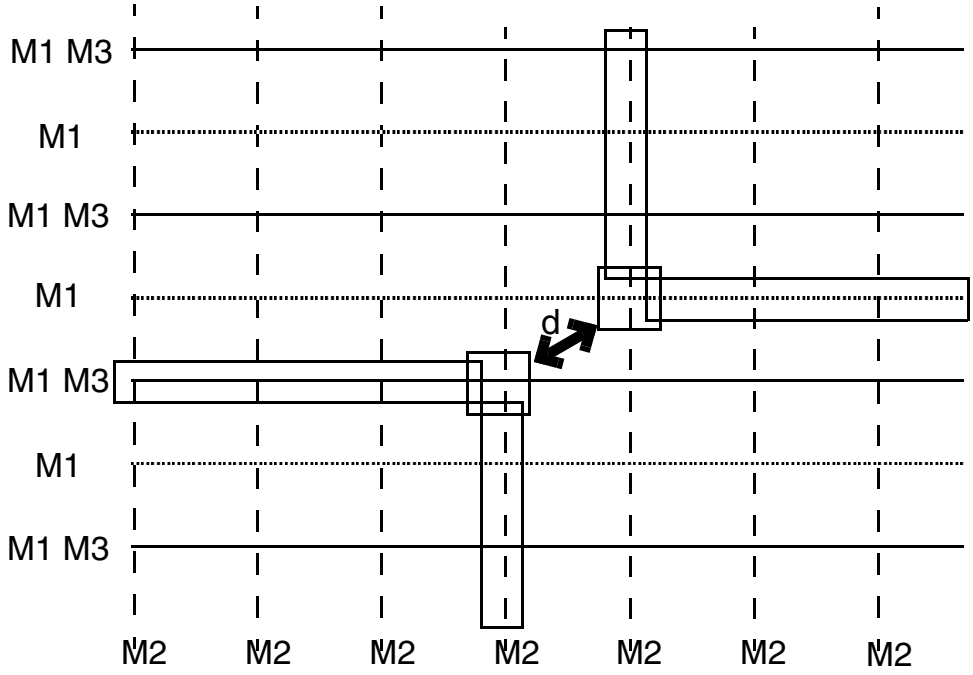
Line-to-Via Spacing: The line-to-via dimension is the distance that the routing pitch requires in order to legally place a via on the track next to a wire (line). This is illustrated in the figure below.



© 2023 Cadence Design Systems, Inc.

Adjusting the Routing Pitch: Most processes allow diagonal vias when the routing pitch is set to line-to-via. In some cases it is necessary to adjust the vertical routing pitch to allow diagonal vias. This adjustment is referred to as *expanded line-to-via spacing*.

Expanded line-to-via pin spacing gives better routing density than via-to-via spacing. Use via-to-via spacing only if your process requires large vias. This is illustrated in the figure below.



If diagonal vias are not allowed when using the line-to-via routing pitch, expand the routing pitch just enough to make diagonal vias allowed. The amount that the pitch must be expanded depends on the cap size of the vias.

When diagonal vias are placed, the spacing (d) must be greater than the *M1*-to-*M1*, *M2*-to-*M2*, and *M3*-to-*M3* spacing rule, depending on the layers in the via.

Related Topics

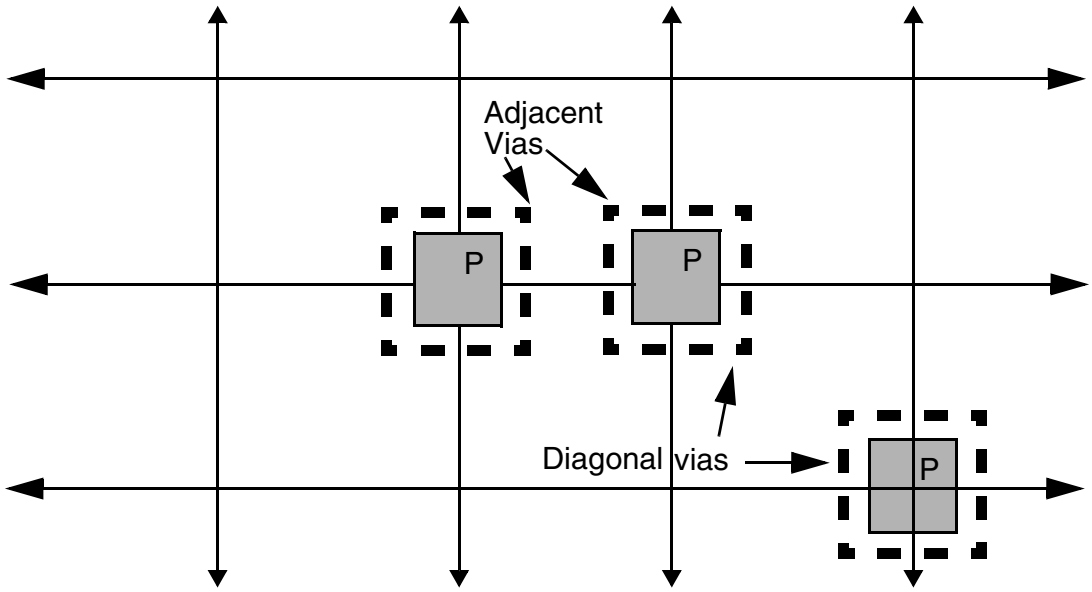
Guidelines for Setting Cell Porosity and High Density in a Cell Library

© 2023 Cadence Design Systems, Inc.

Pin Spacing in Abstract Generator

Pin spacing is usually more than line-to-via spacing and less than via-to-via spacing. Start with line-to-via spacing and, if necessary, increase the pin spacing to allow access to both pins by dropping vias (adjacent vias). Normally, the maximum spacing necessary is via-to-via spacing. A single grid pin is a pin with only one port that covers only one grid point.

Use pin spacing so that it is legal to drop vias on all pins, as shown in the following figure.



If you place single grid pins on orthogonally adjacent grid points, you *must* use via-to-via routing pitch; otherwise, the router might not be able to access the pins.

If you place single grid pins on diagonally adjacent grid points, your routing pitch *must* be able to accommodate diagonal vias.

Related Topics

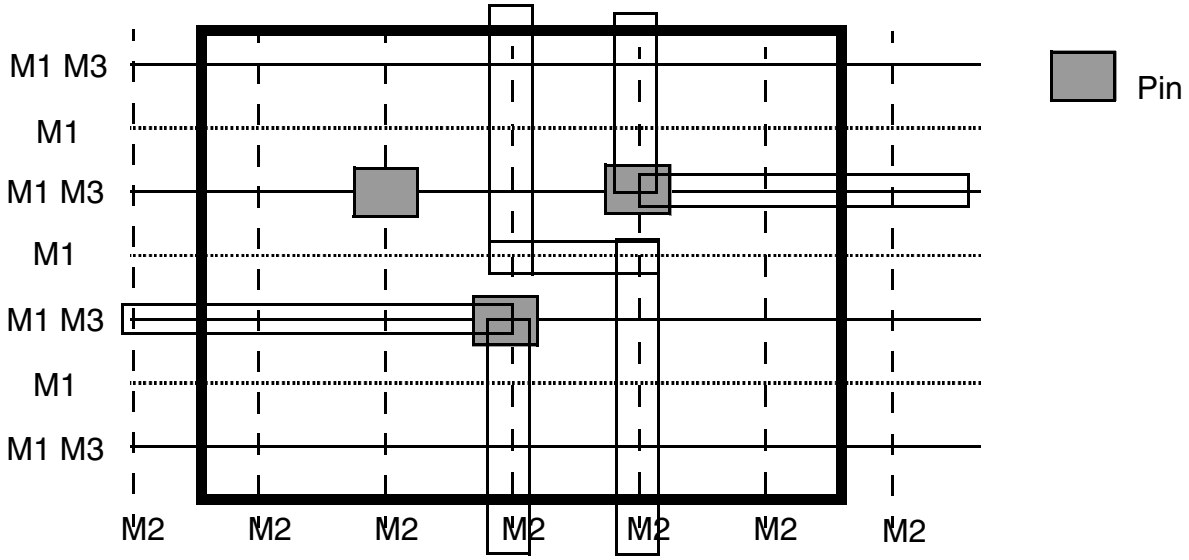
Guidelines for Defining Blockages in a Cell Library

Guidelines for Setting Cell Porosity and High Density in a Cell Library

© 2023 Cadence Design Systems, Inc.

Pin Staggering in Abstract Generator

If you have one vertical routing layer, and line-to-via pin spacing, you should stagger the pin placement, as shown in the following figure.



If pins are at the same y coordinate, there is no legal way to route through the area. However, placing pins at alternate y coordinates allows the final router to route vertically through the cell. (Allow one vertical feed through for every three pins.)

If pins are not staggered as shown, the router can only route vertically between cells. This makes designs with high row utilizations unroutable.

Related Topics

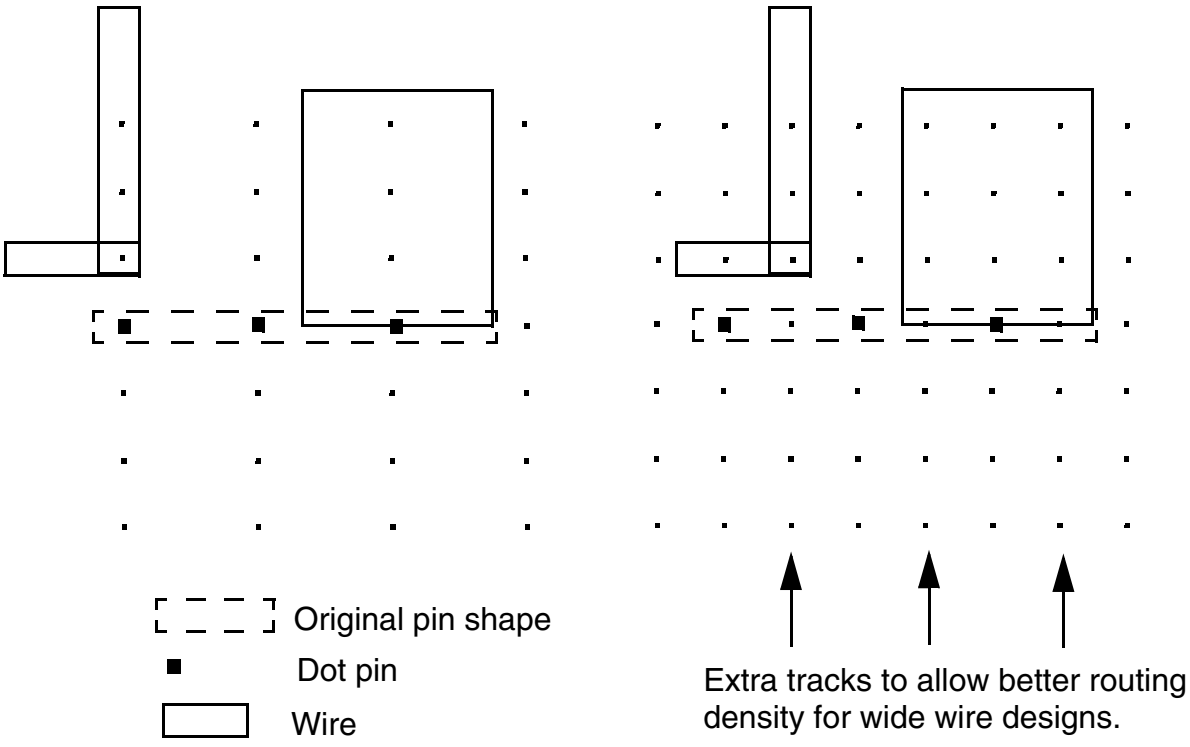
Guidelines for Defining Blockages in a Cell Library

Guidelines for Setting Cell Porosity and High Density in a Cell Library

© 2023 Cadence Design Systems, Inc.

Tracks for Wide Wire Designs

If you route wide and narrow wires, you can get better on-grid routing density by doubling the number of tracks, as shown in the following figure.



For designs with a combination of wide wires and narrow wires, follow these guidelines:

- The design on the left shows the closest that wide wires can be placed using the normal, expanded line-to-via routing pitches.
- The design on the right shows better routing density, if you double the number of tracks.

Related Topics

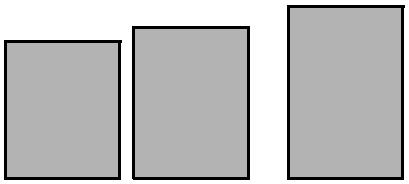
Guidelines for Defining Blockages in a Cell Library

Guidelines for Setting Cell Porosity and High Density in a Cell Library

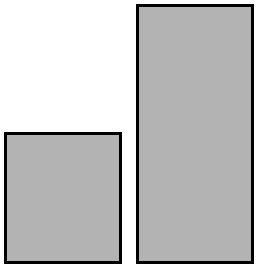
© 2023 Cadence Design Systems, Inc.

Variable-Height and Multiple-Height Cells

Cell heights in your library affect floorplanning and placement. This is shown in the following figure.



Variable-height cells
(Cell heights are not multiples of each other.)



Multiple-height cells
(Single, double, triple, etc.)

See Variable Height Cells and Multiple Height Cells.

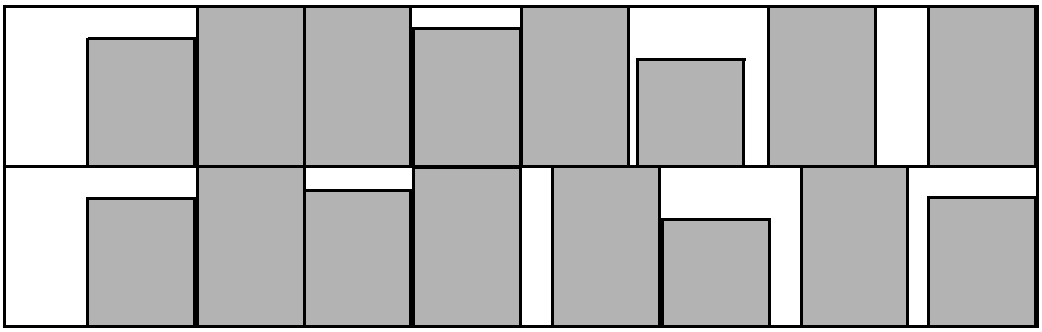
Variable Height Cells

If your design has variable height cells, define only one site type. The following guidelines should be used to assist you.

- The tallest cell's height should be used to define row (site) height.
- Cells are aligned by the lower left corner.
- Power must abut between all cells in the row.
- Placement space is wasted where short cells are placed.
- Floorplanning the core area is simplified because there is only one type of core row.

© 2023 Cadence Design Systems, Inc.

In the following figure, the cell heights are not multiples of each other.



As a result of this design,

- Floorplanning is simplified because additional row types are not added.
- Placement space is wasted because the rows do not overlap.
- The lower left corner of the non rotated cell's place-and-route boundary is used as the alignment point.

If cell heights are multiples of each other (single and double height cells, for example), floorplanning is handled differently.

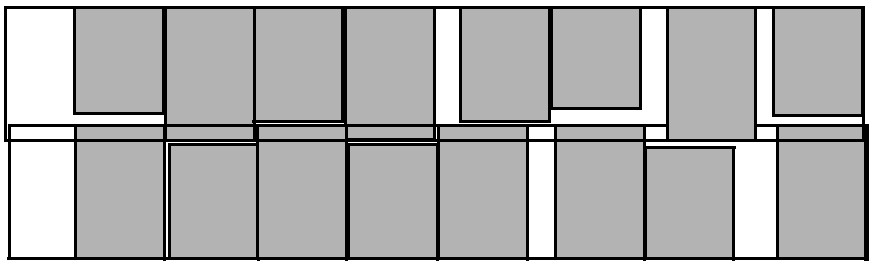
Using the Overlap Layer

You can use the overlap layer to increase placement density and eliminate most of the empty space. The following guidelines should be used to assist you.

- The site for each cell is the same height (use the tallest cell to define the site).
- The overlap layer defines the cell height.
- Empty space above short cells is used by taller cells in the next row.

© 2023 Cadence Design Systems, Inc.

- The placer will resolve overlaps as shown.



If power overlaps:

- The cells must have power pins defined so they connect when cells abut.
- If flipping of rows is allowed, you must define power to accommodate this.

Cell rows can overlap where power does not overlap.

Multiple Height Cells

The following guidelines should be used to assist you when using multiple-height cells.

- These cells have a height that is a multiple of the normal cell height.
- Define a site for each cell height.
- Rows can overlap as long as the power rails align.
- A true channel-less library, where no routing channels are needed between core cell rows, provides best placement.
- Advantages
  - Multiple-height cells utilize placement space better than variable-height cells.
- Disadvantages
  - Overlapping single and multiple-height rows complicates floorplanning.
  - Utilization calculations are more complex.

If a cell is over twice as high, then it limits routing space between the short rows, based on the large cells abutment.

Currently, Innovus does not report utilization values that take into account overlapping rows.

© 2023 Cadence Design Systems, Inc.

Related Topics

Guidelines for Setting Cell Porosity and High Density in a Cell Library

© 2023 Cadence Design Systems, Inc.

Non-rectangular Blocks

If you have non-rectangular blocks, some of the area might be available for standard cell placement.

For an optimal design, follow these guidelines:

- In the LEF file use the overlap layer to describe the area that is not available for placement of other cells.
- Otherwise, by default, the bounding box of the boundary or the `SIZE` statement in the LEF file is used for the shape of the cell.
- Use Abstract Generator to translate the boundary layer into the LEF overlap layer.

Define an blockage on the `OVERLAP` layer in the block (mega cell) over the areas that are not available for cell placement.

Related Topics

Guidelines for Defining Blockages in a Cell Library

Guidelines for Setting Cell Porosity and High Density in a Cell Library

© 2023 Cadence Design Systems, Inc.

E

Technology File Requirements for Abstract Generation

This section describes how Abstract Generator handles technology information in the Virtuoso Studio design environment, outlines the technology information required to run Abstract Generator on OpenAccess, identifies potential technology file problems, and suggests possible solutions to those problems.

You can generate technology information for Abstract Generator, supplement that information by importing LEF, and modify technology information using the Technology File Manager.

Technology information provides details of the process technology used during IC fabrication, including the names of layers, colors and fill patterns, GDSII layer mapping data, and design rules for various layers and vias. Abstract Generator can handle OpenAccess technology information associated with your design library.

You can list the technology information required to run Abstract Generator on OpenAccess. This includes the definition of layers and layer rules, layer functions, physical rules, and vias along with specific requirements for antenna calculation, and viewing data in the Virtuoso Layout Editor.

The following are the three main sources of technology information for a design library in the Virtuoso Studio design environment:

- A binary technology file, called `tech.db`, stored in the library directory
- A separate technology library, which is attached as a property to the design library
- A display resource file, `display.drf`, stored in the library directory

Related Topics

Technology File Requirements for Virtuoso Abstract Generator

Troubleshoot Technology File-Related Issues in the Virtuoso Studio Design Environment

© 2023 Cadence Design Systems, Inc.

Modification of Technology Information

You can modify the technology information for your library to provide all the information Abstract Generator needs.

You can modify the technology information in the following two ways:

- Use *File – Import – LEF* to import additional information into your existing technology file. Any additional information present in the LEF file is added to the existing technology information for the library.

  If you want to replace the existing technology information with the information from a LEF file, select the *Overwrite* option on the *Import LEF* form.

- If you need to make more extensive manual changes to your technology information, use *Technology File Manager* to generate an ASCII version of the technology file. You can edit this file as required and recompile it for use with the library.

  To launch the *Technology File Tool Box*, select *Tools – Technology File Manager* in the CIW.

Consider a situation in which the technology file might need to be updated is when purposes are missing for a routing layer. This is because the technology file must define the LPPs for all routing layers. In case purposes are not defined for a routing layer, Abstract Generator updates the technology file by automatically creating the LPPs. Alternatively, you can update the technology file manually by using the Technology File Manager. Abstract Generator will not proceed if an LPP is missing for a routing layer (ABS-10034).

In both cases where either you update the technology file or Abstract Generator does it automatically, the technology file should be writable (ABS-10050).

Abstract Generator allows you to open a library and proceed with the Abstract Generator flow if the pin and net are not defined for masterslice or cut layers. Abstract Generator issues warning messages for the missing purposes on the masterslice or cut layers. The drawing purpose does need to be defined for routing as well as masterslice and cut layers. However, if during the abstract generation flow, a pin or a blockage is created on a masterslice or cut layer that does not have a corresponding purpose defined in the technology file, Abstract Generator generates one of the following errors:

- ABS-12024: Cell name: Trying to create a pin shape on layer layerName. This layer does not have pin purpose in the technology data.
- ABS-13022: Cell name: Trying to create a pin shape on layer layerName. This layer does not have net purpose in the technology data.

© 2023 Cadence Design Systems, Inc.

- ABS-14029: Cell name: Trying to create a blockage on layer layerName. This layer does not have boundary purpose in the technology data.

Related Topics

Technology File Manager and technology files

© 2023 Cadence Design Systems, Inc.

Technology File Requirements for Virtuoso Abstract Generator

This section describes the technology information required for Abstract Generator to run in the Virtuoso Studio design environment.

- Devices
- Layer Definitions
- Via Definitions
- Physical Rules
- Voltage Dependent and Layer-Purpose-Based Rules

In addition to the shape, Abstract Generator respects the `minArea` and `minRectArea` technology file constraints while creating pins. The minArea constraint defines the minimum area for all pins; whereas the `minRectArea` constraint defines the minimum area for only rectangular-shaped pins.

When both constraints are defined, then the `minRectArea` constraint applies to all rectangular pins, whereas the `minArea` constraint applies to all non-rectangular pins.

If the `minRectArea` constraint is not defined, then the `minArea` constraint applies to both, rectangular as well as non-rectangular pins.

If Abstract Generator fails to attach the '%s' technology library to the '%s' name library. Ensure the following:

1. Use the `absAttachTechLib` command after setting a library. You can do this only in `nogui` or `replay` modes.
2. You have permission to write in the design library, specifically the prop.xx file, which contains property information regarding the attached technology library.
3. The design library exists and that it is not locked.
4. The technology library you are trying to attach exists.

Devices

The device definitions in OpenAccess 2.2 are translated to via definitions.

© 2023 Cadence Design Systems, Inc.

Via Cellviews in OpenAccess

In OpenAccess, the `lef2oa` translator generates a via as a layout cellview with the view name `via` and includes it in the library; For example,

library/*viaName*/via/layout.oa

It also adds a corresponding `customViaDef` to the technology file, which refers to the via cellview.

Layer Definitions

The table below lists the required subsections from the `layerDefinitions` section.

Subsection	Description	Example

techLayers	Defines the layers to be used in your design.
	
	This layer list must include two or more metal layers and one or more via layers. Each layer must have a name, a number, and an abbreviation for the layer name.
		`layerDefinitions(`
		` techLayers(`
		
		` ;( LayerName Layer# Abbreviation )`
		` ;( --------- ------ ------------ )`
		` ;  User-Defined Layers:`
		`  .  .`
		`  ( ndiff  1  ND  )`
		`  ( pdiff  2  PD  )`
		`  ( pwell  13  PW  )`
		`  .  .`
		` ) ;techLayers`
		`) ;layerDefinitions`

© 2023 Cadence Design Systems, Inc.

Subsection	Description	Example

techLayerPurposePriorities	Defines the display priority order of layer purpose pairs (LPPs).
	
	Lists the layer-purpose pairs (LPPs) in the order in which they are to be displayed; applications display an LPP on top of the one that precedes it in the list and underneath the one that follows it.
	
	All routing layers must have the purposes net, pin, and drawing. Abstract Generator adds these automatically if required.
		`layerDefinitions(`
		` techLayerPurposePriorities(`
		
		` ;layers are ordered from the lowest`
		` ;to the highest priority`
		
		` ;( LayerName Purpose )`
		` ;( --------- -------- )`
		`  ( nwell drawing )`
		`  ( nwell pin )`
		`  ( nwell net )`
		`  ( nwell boundary )`
		
		` ) ;techLayerPurposePriorities`
		`) ;layerDefinitions`

layerRules		`layerRules(`
		` functions(`
		
		`  ;( layer function [maskNumber])`
		`  ;( ----- -------- ------------)`
		
		`  ( metal1 "metal" 1 )`
		`  ( metal2 "metal" 2 )`
		
		` ) ;functions`
		`) ;layerRules`

© 2023 Cadence Design Systems, Inc.

Subsection	Description	Example
LEFDefaultRouteSpec		constraintGroups(("LEFDefaultRouteSpec" ; layer constraints interconnect((validLayers (METAL1 METAL2 METAL3)))) ;interconnect) ;LEFDefaultRouteSpec) ;constraintGroups

© 2023 Cadence Design Systems, Inc.

Layer Names

You can encounter the following scenarios while defining layer names:

Scenario 1: If the layer name has not been defined in all the required technology file classes:

- Ensure that the layer is defined in the *techLayers* and *techLayerPurposePriorities* sections of the technology file.
- Ensure that it is listed in the *LEFDefaultRouteSpec* and that it has layer function metal in the *layerfunctions* section.

Use the Technology File Manager to ensure that the layer has been defined in all the sections listed below.

- `layerRules`

```
layerRules(
 functions(

  ;( layer     function   [maskNumber])
  ;( -----     --------   ------------)
   ( metal1    "metal"      1          )
   ( metal2    "metal"      2          )

 );functions
);layerRules
```

- `LEFDefaultRouteSpec`

```
constraintGroups(
 ( "LEFDefaultRouteSpec"
   ; layer constraints
   interconnect(
     ( validLayers ( METAL1 METAL2 METAL3 )   )
   ) ;interconnect
 ) ;LEFDefaultRouteSpec
) ;constraintGroups
```

- `techLayers`

```
layerDefinitions(
 techLayers(
  ;( LayerName             Layer#     Abbreviation )
  ;( ---------             ------     ------------ )
  ;User-Defined Layers:
   ( metal1                23         metal1       )
   ( metal2                27         metal2       )
 ) ;techLayers
) ;layerDefinitions
```

- `techLayerPurposePriorities`

```
layerDefinitions(
 techLayerPurposePriorities(
  ;layers are ordered from lowest to highest
  ;priority
```


© 2023 Cadence Design Systems, Inc.

```
  ;( LayerName                  Purpose    )
  ;( ---------                  -------    )
   ( metal1                     drawing    )
   ( metal1                     pin        )
   ( metal1                     net        )
   ( metal1                     boundary   )
   ( metal1                     label      )
   ( metal2                     drawing    )
   ( metal2                     pin        )
   ( metal2                     net        )
   ( metal2                     boundary   )
   ( metal2                     label      )
 );techLayerPurposePriorities
);layerDefinitions
```

Use the Technology File Manager to define all the required via layers in the `standardViaDefs` section. For any via layer, `layer1` is always the routing layer below and `layer2` is always the routing layer above; for example,

```
        viaDefs(
         standardViaDefs(
         ;(viaDefName layer1 layer2 (cutLayer cutWidth
         ;cutHeight [resistancePerCut])
         ;(cutRows cutCol (cutSpace))
         ;(layer1Enc) (layer2Enc) (layer1Offset)
         ;(layer2Offset) (origOffset)
         ;[implant1 (implant1Enc) [implant2
         ;(implant2Enc)]])
         ;( ------------------------------------------ ------------------------
        ------)

          ( VIA12  METAL1  METAL2  ("VIA1" 0.2 0.2)
           (1 1 (0.0 0.0))
           (0.1 0.1) (0.2 0.2) (0.0 0.0) (0.0 0.0)
           (0.0 0.0)
          )
         ) ;standardViaDefs
        ) ;viaDefs
```

Additionally, you must list the via layer in the `functions` subclass of the `layerRules` section and ensure that it has the function `cut`; for example,

```
layerRules
 functions(
  ;( layer   function   [maskNumber])
  ;( -----   --------   ------------)

   ( Via1    "cut"       5          )
 ) ;layerFunctions
) ;layerRules
```

Scenario 2: If you fail to initialize the technology file because the layer does not have the purpose defined.

© 2023 Cadence Design Systems, Inc.

1. Define the missing layer property in the *techLayerPurposePriorities* section of the technology file and reattach the technology file.

   If you want Abstract Generator to extract through a particular layer, the layer in question must have purposes `pin`, `drawing`, and `net` defined for it in the technology file.

2. You can use the Technology File Manager to define the missing layer property in the `techLayerPurposePriorities` section of the technology file; for example,

```
techLayerPurposePriorities(
 ;layers are ordered from lowest to highest priority
 ;( LayerName                 Purpose    )
 ;( ---------                 -------    )

  ( metal1                    drawing    )
  ( metal1                    pin        )
  ( metal1                    net        )
  ( metal1                    boundary   )
 ) ;techLayerPurposePriorities
```

Scenario 3: The layer definition is missing from one or more of the sections in the technology file

If the layer is defined in the *spacings* section of the *foundry* constraint group, and its definition is missing from one or more of these sections in the technology file: *techLayers, techLayerPurposePriorities, techDisplays; functions* subsection in the *layerRules* section; or *interconnect* section of the *LefDefaultRouteSpec* constraint group. You can update the technology file and try again.

Use the Technology File Manager to define the layer in all the required sections in the technology file. Save and reload the file. A layer needs to be defined in all the following sections of the technology file:

- `techLayers`, `techLayerPurposePriorities`, and `techDisplays`
- `functions` subclass in the `layerRules` section
- `interconnect` constraint category in the `LefDefaultRouteSpec` constraint group
- `spacings` constraint category in the `foundry` constraint group section

  A layer is regarded as not fully defined if it is not defined in any of these sections.

If the layer is also defined in the `techDerivedLayers` section that is used for defining defined layers and is not defined in the `functions` subsection of the `layerRules` section, Abstract Generator issues the info message “Abstract Generator does not support derived layers and proceeds with further execution”. You should evaluate this info message carefully because Abstract Generator does not support derived layers. If such a layer is encountered,

© 2023 Cadence Design Systems, Inc.

Abstract Generator will completely ignore the layer; existing shapes on this layer will not be processed and no shapes will be added on this layer.

Scenario 4: The layer name spacing value is not a multiple of the current manufacturing grid value,

The `width`, `pitch`, `separation`, and `offset` values of all metal layers must be a multiple of the current manufacturing grid value. Update the spacing value for this layer in the *spacing* section of the *foundry* constraint group and the *routingGrid* section of the *LefDefaultRouteSpec* constraint group in the technology file.

Use the Technology File Manager to correct the values.

- The manufacturing grid is set in the controls section. The width and spacing values are set in the `spacings` constraint category of the `foundry` constraint group; for example,

```
controls(
  mfgGridResolution(
  ( 0.005000 )
  ) ;mfgGridResolution
) ;controls

constraintGroups(
 ( "foundry"
   ;physical constraints
   spacings(

   ;( constraint layer1 layer2 value )
   ;( ---------- ------ ------ ----- )

    ( minWidth   "metal1"      0.5 )
    ( minSpacing "metal1"      0.5 )
    ( minWidth   "metal2"      0.5 )
    ( minSpacing "metal2"      0.5 )
   ) ;spacings
 ) ;foundry
) ;constraintGroups
```

- The routing pitch and routing offset values are set in the `routingGrids` constraint category `LEFDefaultRouteSpec` constraint group; for example,

```
constraintGroups(
 ( "LEFDefaultRouteSpec"
   ; physical constraints
   routingGrids(
     ( rightDiagPitch   METAL5    0.01 )
     ( rightDiagOffset  METAL5    0.05 )
     ( leftDiagPitch    METAL5    0.2  )
     ( leftDiagOffset   METAL5    0.4  )
   ) ;routingGrids
 ) ;LEFDefaultRouteSpec
) ;constraintGroups
```

Scenario 5: The '%s' layer does not have a minimum width. Define width in the *spacings* section of *foundry* constraint group of the technology file and then try again.

© 2023 Cadence Design Systems, Inc.

Use the Technology File Manager to define the minimum width for a layer in the `spacings` section of `foundry` constraint group; for example,

```
constraintGroups(
 ( "foundry"
   ;physical constraints
   spacings(

   ;( constraint layer1 layer2 value )
   ;( ---------- ------ ------ ----- )

    ( minWidth   "metal1"      0.5 )
   ) ;spacings
 ) ;foundry
) ;constraintGroups
```

Scenario 5: Unable to initialize Abstract Generator because either the name layer for the name via is not correctly defined.

(ABS-10048)The layer is either missing from `standardViaDefs` section or is not defined in the `techLayers` section. Information about via definitions is included in `standardViaDefs` in the `viaDefs` section.

Ensure that the layer is correctly specified in the *standardViaDefs* subsection of the *ViaDefs* section and in the *techLayers* subsection of the *layerDefinitions* section in the technology file and then try again. To do this:

1. Ensure that the layer is listed correctly in the `standardViaDefs` section; for example,

```
viaDefs(
 standardViaDefs(
 ;(viaDefName layer1 layer2 (cutLayer cutWidth
 ;cutHeight [resistancePerCut])
 ;(cutRows cutCol (cutSpace))
 ;(layer1Enc) (layer2Enc) (layer1Offset)
 ;(layer2Offset) (origOffset)
 ;[implant1 (implant1Enc) [implant2
 ;(implant2Enc)]])
 ;( ------------------------------------------- -----------------------
------)

  ( VIA12  METAL1  METAL2  ("VIA1" 0.2 0.2)
   (1 1 (0.0 0.0))
   (0.1 0.1) (0.2 0.2) (0.0 0.0) (0.0 0.0)
   (0.0 0.0)
  )
 ) ;standardViaDefs
) ;viaDefs
```

   Note: For any via layer, `layer1` is always the lower routing layer and `layer2` is always the upper routing layer.

2. Ensure that all the layers listed are defined in the `techLayers` section; for example,

```
layerDefinitions(
 techLayers(
  ;( LayerName             Layer#     Abbreviation )
```


© 2023 Cadence Design Systems, Inc.

```
    ;( ---------              ------     ------------ )
    ;User-Defined Layers:
     ( metal1                 23         metal1       )
     ( v1                     25         v1           )
     ( metal2                 27         metal1      )
     ( v2                      32         v1          )    ( metal3
    34         metal1       )
     ( v3                     35          v1           )    ( metal4
     36         metal1       )
     ( v4                     52          v1           )    ( metal5
     53         metal1       )
   ) ;techLayers
  ) ;layerDefinitions
```

Scenario 5: Failed to create the name, purpose layer-purpose pair in the technology file.

(ABS-10050) Ensure that you have permissions to write to the technology file and then try again. To do this:

1. Specify the layer-purpose pair information in the `techPurposes` construct in the `layerDefinitions` section during the abstract generation process.

   If missing, you can append this information to the technology file.
2. If write permissions exist on the technology file, Abstract Generator will automatically add the missing layer-purpose pair. However, you would not be able to update the technology file if you do not have the write permission on the file, which is when Abstract Generator issues this message.
3. Change the permission setting of the technology file or contact the person responsible for updating the technology file.
4. Use the Technology File Manager to add the missing layer-purpose definition to the technology file prior to invoking Abstract Generator.

© 2023 Cadence Design Systems, Inc.

Via Definitions

A via layer connects two conducting layers.

On OpenAccess 2.2, information about the via layers is determined from the `viaDefs` section of the technology file. There are two types of `viaDefs` — `standardViaDefs` and `customViaDefs`.

Standard Vias

The `standardViaDefs` has a unique name and is associated with two layers and a list of via parameters with default values.

Example

```
viaDefs(
 standardViaDefs(

 ;(viaDefName layer1 layer2 (cutLayer cutWidth cutHeight [resistancePerCut])
 ;(cutRows cutCol (cutSpace))
 ;(layer1Enc) (layer2Enc) (layer1Offset) (layer2Offset) (origOffset)
 ;[implant1 (implant1Enc) [implant2 (implant2Enc)]])
 ;(------------------------------------------------------------------------)

  ( via1Array METAL1 METAL2 ("VIA12"   0.36     0.36)
  (1 1     (0.36    0.36))
  (0.0 0.0)    (0.0 0.0)     (0.0 0.0)      (0.0 0.0)        (0.0 0.0)
  )
 ) ;standardViaDefs
) ;viaDefs
```

Custom Vias

The `customViaDefs` has a unique name and is associated with a master cellview and two layers. A `customViaDef` is a special purpose via. To specify a custom via, the cellview on which it is based must exist in the specified library. See viaDefs.

Example

```
viaDefs(
 customViaDefs(
 ;(viaDefName libName cellName viewName layer1     layer2  resistancePerCut)
 ;(---------- ------- -------- -------- ------     ------  ----------------)

  ( M2_M1_Via1 lib     M2_M1    via      METAL1     METAL2  4.0 )
  ( M1_Poly    lib     M1_Poly  via      POLY       METAL1  4.0 )
 ) ;customViaDefs
) ;viaDefs
```

Use the Technology File Manager to define all the required via layers in the `standardViaDefs` section. For any via layer, `layer1` is always the routing layer below

© 2023 Cadence Design Systems, Inc.

and `layer2` is always the routing layer above; for example,

```
viaDefs(
 standardViaDefs(
 ;(viaDefName layer1 layer2 (cutLayer cutWidth  ;cutHeight [resistancePerCut])
 ;(cutRows cutCol (cutSpace))
 ;(layer1Enc) (layer2Enc) (layer1Offset)
 ;(layer2Offset) (origOffset)
 ;[implant1 (implant1Enc) [implant2
 ;(implant2Enc)]])
 ;( ------------------------------------------ ------------------------------)

  ( VIA12  METAL1  METAL2  ("VIA1" 0.2 0.2)
   (1 1 (0.0 0.0))
   (0.1 0.1) (0.2 0.2) (0.0 0.0) (0.0 0.0)
   (0.0 0.0)
  )
 ) ;standardViaDefs
) ;viaDefs
```

Additionally, you must list the via layer in the `functions` subclass of the `layerRules` section and ensure that it has the function `cut`; for example,

```
layerRules
 functions(
  ;( layer   function   [maskNumber])
  ;( -----   --------   ------------)

   ( Via1    "cut"       5          )
 ) ;layerFunctions
) ;layerRules
```

Physical Rules

The `spacingRules` subclass included in the `physicalRules` class in the OpenAccess 2.0 technology file is mapped to the `spacings` subsection in the `foundry` constraint group in the OpenAccess 2.2 technology file.

© 2023 Cadence Design Systems, Inc.

In addition to the `spacings` section, Abstract Generator refers to the `spacingTables` section, also in the `foundry` constraint group.

Constraint Category	Description	Example

`spacings`	Specifies the spacing rules, defining the amount of enclosure required when two layers overlap.
	
	All routing layers must have `minWidth` and `minSpacing` values specified.
	
	The `minWidth` constraint specifies the minimum width for any shape on the layer. The `minSpacing` constraint specifies the minimum spacing between objects on a single layer.
		`constraintGroups(`
		`  ( "foundry"`
		`    ;physical constraints`
		`    spacings(`
		
		`    ;( constraint layer1 layer2 value )`
		`    ;( ---------- ------ ------ ----- )`
		
		`     ( minWidth   "Metal1"      0.3 )`
		`     ( minSpacing "Metal1"      0.3 )`
		`    ) ;spacings`
		`  ) ;foundry`
		`) ;constraintGroups`

`spacingTables`	Defines the lookup table(s) for the voltage-based rules, layer-purpose rules, and layer-spacing rules. Abstract Generator looks up the **`spacingTables`** section for these rules to determine the maximum spacing value to be used for cover blockages and for pin cutouts.	`constraintGroups(`
		`  ( "foundry"`
		`    spacingTables(`
		
		`   ;( constraint layer1 [layer2]`
		`   ;((index1Definitions`
		`   ;[index2Defintions])`
		`   ;[defaultValue])`
		`   ;(table) )`
		
		`   ( minSpacing  METAL1`
		`     (( "width" nil nil ))`
		`     (`
		`        0.5 0.333`
		`        0.78 0.99`
		`     )`
		`    ) ;spacingTables`
		`  ) ;foundry`
		`) ;constraintGroups`

© 2023 Cadence Design Systems, Inc.

Constraint Category	Description	Example

controls	Sets the manufacturing grid value in the controls section if no manufacturing grid value is specified in the technology file.
	
	Choose File – Import – LEF to import additional information into your existing technology file. You can use the default value to define the manufacturing grid value in the physicalRules section of the technology file. If you do not specify a default manufacturing grid value in the technology file, Abstract Generator uses a default value of 0.005.
		`controls(`
		`  mfgGridResolution(`
		`  ( 0.005000 )`
		`  ) ;mfgGridResolution`
		`) ;controls`

Abstract Generator does not recognize spacing rules that are defined using `LEF58` constructs. For example the following is an incorrect definition:

```
PROPERTY LEF58_SPACING "
    SPACING 0.077 NOTCHLENGTH 0.140
CONCAVEENDS 0.037 ;" ;
```

The following definition is correct and is recognized by Abstract Generator:

```
LAYER M1
    TYPE ROUTING ;
    DIRECTION VERTICAL ;
    PITCH 0.090000 0.064000 ;
    OFFSET 0.045000 0.000000 ;
    MINWIDTH 0.032 ;
    MAXWIDTH 2 ;
    WIDTH 0.032 ;
SPACING 0.077 NOTCHLENGTH 0.140 ;
```


© 2023 Cadence Design Systems, Inc.

```
END M1
```


© 2023 Cadence Design Systems, Inc.

Voltage Dependent and Layer-Purpose-Based Rules

In addition to the layer rules, such as **minSpacing** and **minWidth**, Abstract Generator now supports the voltage dependent and the layer-purpose-based spacing rules, such as **minVoltageSpacing** (voltage dependent rule) and **minSpacing** (layer-purpose-based). These spacing rules are defined in the **spacings** and **spacingTables** section of the technology file. While loading a library, Abstract Generator reads the rules defined in the technology file and use the read rules for generating abstracts.

The option that enables Abstract Generator to read the voltage-based and layer-purpose-based rules is **ReadVoltageAndPurposeRules**. To enable reading of rules, this option should be set to **TRUE**. By default, this option is set to **FALSE**.

```
absSetOption(”ReadVoltageAndPurposeRules” “TRUE/FALSE”)
```

Abstract generator conforms to the voltage and layer-purpose-based spacing rules while it calculates the spacing value for cover blockage and pin cutouts. The spacing value used for cover blockage is the maximum value that is available from the voltage dependent, layer-purpose-based, and layer-based spacing rules. The same maximum value is used to cut the spacing around pins.

An example of a voltage dependent and layer-purpose-based spacing rule is given as follows.

```
spacingTables(
    ;(constraintlayer1layer2
    ;   ((index1Definitions[index2Definitions)[defaultValue])
    ;   (table))
    ;(---------------------------------------)
     (minVoltageSpacing         "Metal1"
      (("voltage" nil  nil))
      (
       2.03.0
       5.03.5
     )
    )
     (minVoltageSpacing         "Metal2"
      (("voltage" nil  nil))
      (
       2.04.0
       5.04.5
      )
     )
    );spacingTables
spacings(
```


© 2023 Cadence Design Systems, Inc.

```
        ;(minSpacing               ("Metal1" "drawing")3.2)
         (minSpacing               ("Metal1" "pin")    3.4)
         (minSpacing               ("Metal2" "drawing")3.6)
         (minSpacing               ("Metal2" "pin")    3.8)
       );spacings
```

In the above example, the maximum value of the **minVoltageSpacing** rule for Metal1 layer is **3.5**. The **minSpacing** value of Metal1 layer for purpose **drawing** and **pin** is **3.2** and **3.4**, respectively. Therefore, the values that are considered to calculate the **minSpacing** value are **3.2**, **3.4**, and **3.5**. Because **3.5** is the maximum value that is available from the voltage dependent and layer-purpose-based spacing rules for the Metal1 layer, the **minSpacing** value used for cover blockage and pin cutout is **3.5**.

If the entries for minSpacing table are either non-applicable or the default value is not given. Define appropriately in the spacings section of foundry constraint group of the technology file and then try again. Abstract Generator stops because the minSpacing value set for the given layer is 0.

Use the Technology File Manager to set a valid minSpacing value for the layer; for example,

```
constraintGroups(
 ( "foundry"
   ;physical constraints
   spacings(
   ;( constraint layer1 layer2 value )
   ;( ---------- ------ ------ ----- )
    ( minWidth   "metal1"      0.5 )
   ) ;spacings
 ) ;foundry
) ;constraintGroups
```

Abstract Generator uses this value for the minimum spacing for both metal and via layers.

Related Topics

Technology File Structure

© 2023 Cadence Design Systems, Inc.

Defining a Default Via in the OpenAccess Technology File

Ensure that the technology file includes a fully defined via layer for the layer.

To correctly define a default via in the OpenAccess technology file:

- Define the via in the `customViaDefs` section of the technology file; for example,

```
viaDefs(
 customViaDefs(
  ;(viaDefName libName cellName viewName
  ; layer1     layer2  resistancePerCut)

   ( M2_M1      lib     M2_M1    via
     metal1     metal2  4.0 )
 ) ;customViaDefs
) ;viaDefs
```

  The cellview used in the definition must exist in the technology library. Symbolic and layout cellviews usually correspond to a device defined in the `devices` section. If the cellview does not exist, you can generate one by adding a device to the `devices` section. If you want to use a via cellview, you need to create it manually or use `lef2oa`, which will create it automatically.

- Ensure that the via is listed in the `LEFDefaultRouteSpec`; for example,

```
constraintGroups(
 ( "LEFDefaultRouteSpec"
   ; layer constraints
   interconnect(
     ( validVias ( M1_M2 M2_M1 )   )
   ) ;interconnect
 ) ;LEFDefaultRouteSpec
) ;constraintGroups
```

- List the via layer in the `functions` subclass of the `layerRules` section and ensure that it has function `cut`; for example,

```
layerRules
 functions(
  ;( layer   function   [maskNumber])
  ;( -----   --------   ------------)

   ( v1      "cut"       5          )
 ) ;layerFunctions
) ;layerRules
```

Related Topics

ABS-10030

© 2023 Cadence Design Systems, Inc.

Requirements for Antenna Calculation

This section describes the technology file requirements for using the antenna calculation capabilities of Abstract Generator.

In the technology file on OpenAccess 2.2, the antenna rules have been moved to the `antennaModels` section of the `foundry` constraint group. You specify a separate `antennaModels` group for each oxide type and can specify `antennaModels` for up to four oxide types.

If an `alternate foundry` constraint group (AFCG) is defined in the `.cdsenv` file, Abstract Generator replaces the `foundry` constraint group and uses the specified AFCG instead.

You can specify an AFCG by setting the following environment variable in your `.cdsenv` file:

```
cdba.layout AlternateFoundryCG string "AFCG_name"
```

The constraint specifies the antenna ratios for one oxide type corresponding to a specific thickness of the gate oxide. You can specify the thickness value in the `techLayerProperties` subsection in the `layerDefinitions` section if antenna side area calculation is to be performed.

Note: In OpenAccess, you must specify either side antenna values or non-side antenna values. OpenAccess supports only one of them at a time.

During the Extract step, the following factors are supported while calculating the antenna value. These factors are defined in the `antennaModels` section of the foundry constraint group in the technology file.

- (antennaCutFactor) cumulativeRoutingPlusCut

  If this factor is found in the technology file, the values of the cumulative antenna ratio for `CUT` layers are dumped in the LEF file. In addition, the values of the cumulative antenna ratio for `CUT` layers are added to the cumulative antenna ratio values of the metal layers. This factor has a boolean value. By default, this factor is set to `nil`.

- (antennaMetalFactor) diffArea

  If this factor is found in the technology file, Abstract Generator uses the value of this factor to calculate the cumulative antenna ratio values for the metal layers. However, if this factor is not found in the technology file, Abstract Generator sets it to the default value 1.0 and calculates the cumulative antenna ratio values. This factor has float values.

- (antennaDiffPlusFactor) diffPlus

  If this factor is found in the technology file, Abstract Generator uses the value of this factor to calculate the cumulative antenna ratio values for the metal layers. However, if

© 2023 Cadence Design Systems, Inc.

this factor is not found in the technology file, Abstract Generator sets it to the default value 0.0 and calculates the cumulative antenna ratio values. This factor has float values.

- (antennaDiffMinusFactor)) diffMinus

  If this factor is found in the technology file, Abstract Generator uses the value of this factor to calculate the cumulative antenna ratio values for the metal layers. However, if this factor is not found in the technology file, Abstract Generator sets it to the default value 0.0 and calculates the cumulative antenna ratio values. This factor has float values.

- (antennaDiffAreaReduceFactor) diffAreaReduce

  If this factor is found in the technology file, Abstract Generator uses the value of this factor to calculate the cumulative antenna ratio values for the metal layers. However, if this factor is not found in the technology file, Abstract Generator sets it to the default value 1.0 and calculates the cumulative antenna ratio values. The values for this factor are given in the form of 1-D table.

The formula used to calculate the cumulative antenna ratio values for the metal layers is as follows:

```
areaCar[j] += (((antennaMetalFactor*cumMetalArea) + (antennaCutFactor*cumCutArea))
     * antennaDiffAreaReduceFactor - diffArea * antennaDiffMinusFactor)/
     (cumGateAreaSharingCharge_[j] + antennaDiffPlusFactor * diffArea);
```

Example

```
constraintGroups(
 ( "foundry"
   antennaModels(
   ;( model
   ;( ----
    ( "default"
      antenna(
      ;( ratioName      layer     value )
      ;( ---------      -----     ----- )
       ( areaRatio      METAL1    0.345                  )
       ( diffAreaRatio  METAL1   ((0.45 0.4)(0.55 0.6))  )
       ( areaRatio      VIA1      0.344                  )
       ( diffAreaRatio  VIA1     ((0.44 0.5)(0.57 0.8))  )
      ) ;antenna

      cumulativeMetalAntenna(
      ;( ratioName         value )
      ;( ---------         ----- )
       ( areaRatio         0.544 'side  )
       ( diffAreaRatio     0.888        )
      ) ;cumulativeMetalAntenna

      cumulativeViaAntenna(
      ;( ratioName         value )
      ;( ---------         ----- )
       ( areaRatio         0.654 'side  )
       ( diffAreaRatio     0.884        )
      ) ;cumulativeViaAntenna
```


© 2023 Cadence Design Systems, Inc.

```
    ) ;default

    ( "second"
      antenna(
      ;( ratioName      layer     value )
      ;( ---------      -----     ----- )
       ( areaRatio      METAL1    1.345 'side               )
       ( diffAreaRatio  METAL1   ((0.455 0.4)(0.555 0.65))  )
      ) ;antenna


antenna(
( areaRatio "poly" <value> )
); antenna

      cumulativeMetalAntenna(
      ;( ratioName         value )
      ;( ---------         ----- )
       ( areaRatio         3.564        )
       ( diffAreaRatio     7.888        )
      ) ;cumulativeMetalAntenna

      cumulativeViaAntenna(
      ;( ratioName         value )
      ;( ---------         ----- )
       ( areaRatio         7.654        )
       ( diffAreaRatio     1.884        )
      ) ;cumulativeViaAntenna
    ) ;second

    ( "third"
      antenna(
      ;( ratioName      layer     value )
      ;( ---------      -----     ----- )
       ( areaRatio      METAL1    0.343                     )
       ( diffAreaRatio  METAL1   ((0.43 0.3)(0.53 0.6))     )
      ) ;antenna

      cumulativeMetalAntenna(
      ;( ratioName         value )
      ;( ---------         ----- )
       ( areaRatio         9.765 'side  )
       ( diffAreaRatio     0.188        )
      ) ;cumulativeMetalAntenna

      cumulativeViaAntenna(
      ;( ratioName         value )
      ;( ---------         ----- )
       ( areaRatio         3.22                                )
       ( diffAreaRatio    ((0.421 0.4)(0.7 0.64)(0.81 0.59))    )
      ) ;cumulativeViaAntenna
    ) ;third

    ( "fourth"
      antenna(
      ;( ratioName      layer     value )
      ;( ---------      -----     ----- )
       ( areaRatio      METAL2    0.645 'side               )
       ( diffAreaRatio  METAL2   ((0.47 0.4)(0.55 0.7))     )
      ) ;antenna
```


© 2023 Cadence Design Systems, Inc.

```
      cumulativeMetalAntenna(
      ;( ratioName         value )
      ;( ---------         ----- )
       ( areaRatio         0.6                          )
       ( diffAreaRatio     ((0.127 0.23)(0.32 0.7))     )
      ) ;cumulativeMetalAntenna

      cumulativeViaAntenna(
      ;( ratioName         value )
      ;( ---------         ----- )
       ( areaRatio         9.67     )
       ( diffAreaRatio     0.684    )
      ) ;cumulativeViaAntenna
    ) ;fourth

      ( "factors"
      ( cumulativeRoutingPlusCut "METAL1" t )
      ( area "METAL1" 0.50 )
      ( diffPlus "METAL1" 0.60 )
      ( diffMinus "METAL1" 0.70 )
      ( diffAreaReduce "METAL1" (
      ( 0.0 20.0 )
      ( 0.5 30.0 )
      ( 1.0 40.0 )
    ))
    ) ;factors
  ) ;antennaModels

   ) ;antennaModels
 ) ;foundry
) ;constraintGroups
```

Related Topics

Technology File Structure

Processing Antenna Information in Standalone Abstract Generator

Calculating and Fixing Process Antenna Violations

© 2023 Cadence Design Systems, Inc.

Requirements for Metal Density Calculation

All layers being abstracted should have a minimum density value specified in the technology file. The maximum density value is al. If a step size exists for both `maxDensity` and `minDensity` constraints, the value should match.

```
constraintGroups(
 ("foundry"
   spacings(
     ("minDensity" tx_layer g_density)
       ...
   ) ;spacings
  ) ;foundry
);constraintGroups

constraintGroups(
 ("foundry"
   spacingTables(
      ("minDensity" tx_layer
      ( ("step" nil nil) [ g_defaultValue ] )
      (g_table) )
   ) ;spacingTables
  ) ;foundry
);constraintGroups
```

Related Topics

Incremental Technology Databases

Technology File Structure

© 2023 Cadence Design Systems, Inc.

Requirements for Viewing Data In Layout

The section describes the technology file requirements for viewing your data using Virtuoso tools.

Display packets are defined in the display resource file and control how layers appear on your monitor and plotting devices. You assign a display packet to a particular layer-purpose pair and specify other layer display attributes in the `techDisplays` subsection of the `layerDefinitions` section.

Related Topics

Incremental Technology Databases

Technology File Structure

© 2023 Cadence Design Systems, Inc.

Requirements for Specifying Place-and-Route Information

On CDB, the `prRules` class defined the rules for place-and-route applications.

The `prRules` class does not exist in the technology file on OpenAccess. Information stored in the `prRules` class of the CDB technology file is stored in different locations in the OpenAccess 2.2 technology file. Many of these PR rules are mapped to the `LEFDefaultRouteSpec` constraint group (or the specified constraint group) in the OpenAccess 2.2 technology file.

The table below summarizes the mapping between CDB and OpenAccess 2.2 for the sections required by the abstract generator:

CDB Technology File	OpenAccess Technology File
`prRoutingLayers` (Specifies the routing direction of layers used for routing)	A routing layer in the OpenAccess 2.2 technology file must be included in the `validLayers` constraint of the `LEFDefaultRouteSpec` constraint group (or the specified constraint group) *and* have an appropriate `functions` subsection (with function `metal` or `poly`) specified in the `layerRules` section. See LEFDefaultRouteSpec, Layer Functions.
`prMastersliceLayers` (Specifies the masterslice layers in order)	An OpenAccess masterslice layer must be included in the `functions` subsection of the `layerRules` section with the function `poly`. See Layer Functions.

LEFDefaultRouteSpec

In OpenAccess 2.2, `routeSpecs` are mapped to constraint groups. One of these constraint groups is `LEFDefaultRouteSpec` (or the specified constraint group), which is supported by the abstract generator.

Example

```
constraintGroups(
 ( "LEFDefaultRouteSpec"
    ;layer constraints
    interconnect(
    ( validLayers    ( METAL1 METAL2 METAL3 )      )
    ( validVias      ( M1_M2 M2_M1 )               )
    ( errorLayers    GATEOXIDE1)
```


© 2023 Cadence Design Systems, Inc.

```
    ) ;interconnect

    ; physical constraints
    spacings(
      ( minWidth                "METAL1"0.6      )
      ( minSpacing              "METAL1"0.4'soft )
      ( minDiagonalWidth        "METAL1"0.3      )
      ( minDiagonalSpacing      "METAL1"0.2      )
      ( minWireExtension        "METAL1" 0.1      )
    ) ;spacings

    orderedSpacings(
      ( minOppExtension        "METAL1"  "VIA1"(0.1 0.2))
    ) ;orderedSpacings

    routingGrids(
     ( horizontalPitch   M1    0.19  )
     ( horizontalOffset  M1    0.095 )
     ( verticalPitch     M1    0.19  )
     ( verticalOffset    M1    0.095 )
    ) ;routingGrids
  ) ;LEFDefaultRouteSpec
) ;constraintGroups
```

Note: The `prMastersliceLayers`, `prRoutingLayers`, `prRules`, and `routeSpecs` constructs have been deprecated and will be removed in an upcoming release. Cadence recommends that you remove them from your technology files.

Layer Functions

Layer functions are defined in the `functions` subsection of the `layerRules` section and are required in OpenAccess to identify the types of layers in the process. If this information is

© 2023 Cadence Design Systems, Inc.

missing for a particular layer, that layer will be missing from any LEF exported by the abstract generator.

Subsection	Description	Example

`functions`	Specifies the layer functions (materials).
	
	Valid function values include `poly`, `diff`, `cut`, `metal`, `ndiff`, `pdiff`, `nplus`, `pplus`, `nwell`, `pwell`, `li`, `unknown`, `recognition`.
	
	Note: `diff` is a new masterslice layer type in OpenAccess 2.2.
		`layerRules(`
		` functions(`
		
		`  ;( layer  function  [maskNumber])`
		`  ;( -----  --------  ------------)`
		
		`  ( POLY  "poly"  1  )`
		`  ( DIFF  "diff"  2  )`
		`  ( CONT  "cut"  3  )`
		`  ( M1  "metal"  4  )`
		`  ( VIA12  "cut"  5  )`
		`  ( NDIFF  "ndiff"  6  )`
		`  ( DIFF2  "pdiff"  7  )`
		`  ( NPLUS  "nplus"  8  )`
		`  ( PPLUS  "pplus"  9  )`
		`  ( NWELL  "nwell"  10  )`
		`  ( PWELL  "pwell"  11  )`
		`  ( LI  "li"  12  )`
		
		` );functions`
		`);layerRules`

Related Topics

Technology File Structure

Troubleshoot Technology File-Related Issues in the Virtuoso Studio Design Environment

© 2023 Cadence Design Systems, Inc.

Troubleshoot Technology File-Related Issues in the Virtuoso Studio Design Environment

This section lists the common problems found in technology files in the Virtuoso Studio design environment and provides information about how to fix these to enable Abstract Generator to run.

Antenna Values Must Not Equal Zero

Abstract Generator checks all antenna values to ensure that their value is not `0`.

Specifying Routing Grids

To enable Abstract to start, it is required to define routing grids in the technology file. If the routing grids are not defined in the technology file, the technology file will not load properly and an error message will be displayed.

Read-Only Technology File in Abstract Generator

By default, Abstract Generator creates boundary, pin, and net layer purpose pairs for the layers for which it does not find boundary, pin, and net purposes in the technology file, and then tries to save these layer purpose pairs in the technology file. Because the technology file is read-only, the layer purpose pairs are not saved and a warning message is displayed.

To enable Abstract Generator to work with the read-only technology file, the `ReadOnlyTechnology` option has been added to Abstract Generator. When this option is set to `TRUE`, Abstract Generator does not try to save to the technology file the layer purpose pairs that it creates.

You can set the `ReadOnlyTechnology` option to TRUE as follows:

```
absSetOption("ReadOnlyTechnology" "true")
```

Layer Not Fully Defined

Abstract Generator requires that all the layers used in the process are fully defined in the appropriate technology file sections. Abstract Generator issues the message ABS-217 if a layer is defined in the `LEFDefaultRouteSpec` (or the specified constraint group) but is not included in the `functions` subsection in the `layerRules` section or if the layer function in the `functions` subsection is not specified as `metal`.

© 2023 Cadence Design Systems, Inc.

Related error message: ABS-10016

Related Topics

Troubleshoot Metal Layer Issues

Troubleshoot Layer Spacing Rules Issues

Troubleshoot Via Layers Issues

Troubleshoot Routing Layer Issues

Troubleshoot Minimum Width Value Issues

Troubleshoot Minimum Spacing Value Issues

DATABASE MICRONS in the Exported LEF File

© 2023 Cadence Design Systems, Inc.

Troubleshoot Metal Layer Issues

Abstract Generator requires that all the layers used in the process are fully defined in the appropriate technology file sections. Abstract Generator issues the message ABS-10016 if a layer is defined in the `LEFDefaultRouteSpec` (or the specified constraint group) but is not included in the `functions` subsection in the `layerRules` section or if the layer function in the `functions` subsection is not specified as `metal`.

Define routing layers in the `validLayers` constraint in the `interconnect` constraint category of the `LEFDefaultRouteSpec` constraint group (or the specified constraint group). In addition, specify an appropriate `functions` type (for example, `metal`) in the `layerRules` section.

ABS-10016

Unable to define layer name in all the required technology file classes. Ensure that the layer is defined in the techLayers and techLayerPurposePriorities sections of the technology file, it is listed in the LEFDefaultRouteSpec, and that it has layer function metal in the Layer Definitions section.

ABS-10013

The *`name`* layer is not defined in the techLayer class of the technology file.

If at least one of the required layers is not defined in the technology file, then you can use the Technology File Manager to define the missing layers in the `techLayers` section of the technology file.

ABS-10015

Unable to find sufficient metal layers defined in the current design. Use the Technology File Manager to define these layers in the validLayers subsection of LEFDefaultRouteSpec constraint group of the technology file. In addition, ensure that these layers are assigned the function metal in the functions subsection of the Layer Definitions section.

ABS-10019

Failed to initialize the technology file because the '%s' layer does not have the '%s' purpose defined. Define the missing layer property in the techLayerPurposePriorities section of the technology file and reattach the technology file.

© 2023 Cadence Design Systems, Inc.

Related Topics

Troubleshoot Layer Spacing Rules Issues

Troubleshoot Via Layers Issues

Troubleshoot Routing Layer Issues

Troubleshoot Minimum Width Value Issues

Troubleshoot Minimum Spacing Value Issues

© 2023 Cadence Design Systems, Inc.

Troubleshoot Layer Spacing Rules Issues

If a layer has spacing rules that are defined in the `spacingTables` section of a constraint group, Abstract Generator proceeds depending on the layer definition in other sections of the technology file.

- If the layer is also defined in the `functions` subsection of the `layerRules` section, it represents an ideal situation and Abstract Generator proceeds without issuing any messages.
- If the layer is not additionally defined in the `functions` subsection of the `layerRules` section, Abstract Generator stops further execution and issues the error message ABS-10021.

ABS-10021

Layer definition is missing from one or more of these sections in the technology file: techLayers, techLayerPurposePriorities, techDisplays; functions subsection in the layerRules section; or interconnect section of the LefDefaultRouteSpec constraint group. Use the Technology File Manager to define the layer in all the required sections in the technology file. Save and reload the file.

ABS-10059

The layer *`layerName`*, defined in the `spacingTables` section of the *foundry* constraint group, is also defined in the *techDerivedLayers* section of the technology file. The abstract generator does not support derived layers and this layer will, therefore, be ignored. As a result, the abstracts generated will not have any shape on this layer.

Related Topics

Troubleshoot Metal Layer Issues

Troubleshoot Via Layers Issues

Troubleshoot Routing Layer Issues

Troubleshoot Minimum Width Value Issues

Troubleshoot Minimum Spacing Value Issues

© 2023 Cadence Design Systems, Inc.

Troubleshoot Via Layers Issues

Abstract Generator issues an error ABS-20007 and ABS-10018 if there are no via layers defined in the technology file.

Via layer information must be defined in `standardViaDefs` in the `viaDefs` section of the technology file. Abstract Generator requires fully-specified interlayer connectivity for all master slice, cut, and routing layers used in the design. When you are specifying up and down layers for a specific via (cut) layer, the lower layer must be specified first, then the via layer, and then the upper layer.

Example

```
viaDefs(
 standardViaDefs(
 ;(viaDefName layer1 layer2 (cutLayer cutWidth  ;cutHeight [resistancePerCut])
 ;(cutRows cutCol (cutSpace))
 ;(layer1Enc) (layer2Enc) (layer1Offset)
 ;(layer2Offset) (origOffset)
 ;[implant1 (implant1Enc) [implant2
 ;(implant2Enc)]])
 ;(------------------------------------------------------------------------)

  ( VIA12  METAL1  METAL2  ("VIA1" 0.2 0.2)
   (1 1 (0.0 0.0))
   (0.1 0.1) (0.2 0.2) (0.0 0.0) (0.0 0.0)
   (0.0 0.0)
  )
  ( VIA23  METAL2  METAL3  ("VIA2" 0.1 0.2)
   (1 1 (0.0 0.0))
   (0.1 0.1) (0.2 0.2) (0.0 0.0) (0.0 0.0)
   (0.0 0.0)
  )
  ( VIA23Array  METAL2  METAL3  ("VIA2" 0.2 0.2)
   (1 1 (0.3 0.3))
   (0.1 0.1) (0.1 0.1) (0.0 0.0) (0.0 0.0)
   (0.0 0.0)
  )
 ) ;standardViaDefs
) ;viaDefs
```

ABS-20007

There are no vias specified in the technology file. Ensure that the selected constraint group has a *ValidVias* section or at least one via is specified in the *standardViaDefs* subsection of the *viaDefs* section in the technology file. In addition, ensure that the *function* argument is set to '*cut*' for the via in the *functions* subsection of the *layerRules* section.

The `standardViaDefs` section of the technology file defines the via layers. Because a design requires at least two metal layers, there must be at least one via layer defined to connect them. Abstract Generator creates extraction connectivity list using `validvias`

© 2023 Cadence Design Systems, Inc.

section.

Additionally, each via layer must also be defined in the `functions` subsection of the `layerRules` section and must have the function `cut`.

ABS-10018

The '`%s`' via layer is not defined in all the required technology file sections. Ensure that the via layer is listed in the functions subsection of the layerRules section with the function set to 'cut' and then try again.

Related Topics

Troubleshoot Metal Layer Issues

Troubleshoot Layer Spacing Rules Issues

Troubleshoot Routing Layer Issues

Troubleshoot Minimum Width Value Issues

Troubleshoot Minimum Spacing Value Issues

© 2023 Cadence Design Systems, Inc.

Troubleshoot Routing Layer Issues

Abstract Generator issues an error ABS-10047 if the upper layer specified in the `standardViaDefs` section is not a routing layer. You must also specify the layers from lowest to highest, otherwise the technology check might indicate that you have the layers incorrectly ordered.

ABS-10047

Failed to initialize Abstract Generator because either the layer2 slot in via layer *`layer1Name`*-*`cutLayerName`*-*`layer2Name`*, which is currently *`layer2Name`*, must be a routing layer, else the layers might appear in the wrong order in LEF.

To resolve this issue:

1. Define the layer in the *validLayers* section of the *LefDefaultRouteSpec* constraint group in the technology file.
2. Use the Technology File Manager to define the layer correctly in the `validLayers` section of `LEFDefaultRouteSpec` constraint group. The layers must be specified from lowest to highest, otherwise the technology checker might indicate that you have the layers in the wrong order.

   Note: If the layers are in the wrong order, you need to define the vias in the bottom-up order; first layer bottom, second layer cut, and third layer top of the via. The lower level layer must be listed first in `standardViaDefs`; for example, `Poly-Cont-Metal1`.

ABS-10046

Abstract Generator failed to initialize because the layer *`name`* in via layer *`layer1Name`*-*`cutLayerName`*-*`layer2Name`* must be a routing or masterslice layer. To resolve this issue:

1. Define the layer in the appropriate section, either *validLayers* section of *LefDefaultRouteSpec* constraint group or *functions* section of *layerRules* section.
2. Use the Technology File Manager to update your technology file accordingly.
3. You must define masterslice layers in the `functions` subsection of the `layerRules` section.
4. Specify the `function` as one of `ndiff`, `nplus`, `nwell`, `pdiff`, `poly`, `pplus`, `pwell`, or `li`.

© 2023 Cadence Design Systems, Inc.

Related Topics

Troubleshoot Metal Layer Issues

Troubleshoot Layer Spacing Rules Issues

Troubleshoot Via Layers Issues

Troubleshoot Minimum Width Value Issues

Troubleshoot Minimum Spacing Value Issues

© 2023 Cadence Design Systems, Inc.

Troubleshoot Minimum Width Value Issues

Abstract Generator checks the `minWidth` values specified for all the metal layers in the design and issues an error message ABS-10023 if the value is undefined or is `0`.

Abstract Generator issues an error message ABS-10022 if the `minWidth` value is not a multiple of the manufacturing grid for a particular layer. This check is performed only if there is a manufacturing grid value defined in the technology file.

ABS-10023

The '%s' layer does not have a minimum width. Define in the spacings section of foundry constraint group of the technology file and then try again.

ABS-10022

Layer name spacing value is not a multiple of the current manufacturing grid value. The width, pitch, separation, and offset values of all metal layers must be a multiple of the current manufacturing grid value. Update the spacing value for this layer in the *spacing* section of the *foundry* constraint group and the *routingGrid* section of the *LefDefaultRouteSpec* constraint group in the technology file.

ABS-10010

Manufacturing grid value is not specified in the technology file for the '%s' value. Define the manufacturing grid value in the Physical Rules section of the technology file before you run the abstract generator.

ABS-10028

The routing offset value for a layer must be equal to that of the equivalent base layer; for example, `metal2` or `metal1`, depending on which layer is being examined. Use the Technology File Manager to define the offset value in the `LEFDefaultRouteSpec` constraint group of the technology file.

Related Topics

Troubleshoot Metal Layer Issues

Troubleshoot Layer Spacing Rules Issues

© 2023 Cadence Design Systems, Inc.

Troubleshoot Via Layers Issues

Troubleshoot Routing Layer Issues

Troubleshoot Minimum Spacing Value Issues

© 2023 Cadence Design Systems, Inc.

Troubleshoot Minimum Spacing Value Issues

The minimum spacing rule defines the minimum spacing required between objects on a single layer and is defined in either the `spacings` or `spacingTables` constraint category of the `foundry` constraint group in the technology file.

Abstract Generator checks the `minSpacing` value for every metal layer and issues an error message ABS-10024 if the value is undefined or is set to `0`. A manufacturing grid check is also performed here.

ABS-10024

The entries for `minSpacing` table are either non-applicable or the default value is not given. Define appropriately in the *spacings* section of *foundry* constraint group of the technology file and then try again.

Abstract Generator stops because the `minSpacing` value set for the given layer is 0.

Use the Technology File Manager to set a valid `minSpacing` value for the layer; for example,

```
constraintGroups(
 ( "foundry"
   ;physical constraints
   spacings(

   ;( constraint layer1 layer2 value )
   ;( ---------- ------ ------ ----- )

    ( minWidth   "metal1"      0.5 )
   ) ;spacings
 ) ;foundry
) ;constraintGroups
```

Abstract Generator uses this value for the minimum spacing for both metal and via layers.

Related Topics

Physical Rules

Troubleshoot Metal Layer Issues

Troubleshoot Layer Spacing Rules Issues

Troubleshoot Routing Layer Issues

Troubleshoot Minimum Width Value Issues

© 2023 Cadence Design Systems, Inc.

Troubleshoot Cut Layer Issues

Abstract Generator checks to ensure that every cut layer used in a default via has both an upper and a lower routing layer. It stops if this check fails and issues the error message ABS-10029 and 10030.

Define vias in the `LEFDefaultRouteSpec` (`validVias`) and in a custom via definition (`customViaDefs`) in the `viaDefs` class.

ABS-10029

Define the via in the following sections in the technology file: viaDefs (`standardViaDefs` or `customViaDefs`) and validVias in the interconnect section of the `LefDefaultRouteSpec` constraint group.

If you define a layer above a routing layer, you must define a via layer to go between them. Abstract Generator cannot extract connectivity without this via layer. Use the Technology File Manager to define the via layer in all the required technology file sections.

Related Topics

DATABASE MICRONS in the Exported LEF File

Troubleshoot Metal Layer Issues

Troubleshoot Layer Spacing Rules Issues

Troubleshoot Via Layers Issues

Troubleshoot Routing Layer Issues

Troubleshoot Minimum Width Value Issues

Troubleshoot Minimum Spacing Value Issues

© 2023 Cadence Design Systems, Inc.

DATABASE MICRONS in the Exported LEF File

You can use one of the following methods to set the correct `DATABASE MICRONS` values in the LEF file exported from Abstract Generator:

- Import a technology LEF file to create a technology file

  If the imported technology file contains the following statement:

  ```
  UNITS
      DATABASE MICRONS 1000  ;
  END UNITS
  ```

  it will be used by Abstract Generator and will be output to LEF created by Abstract Generator. In OpenAccess, `DATABASE MICRONS` is output to the technology LEF only.

- Manually edit the value in the technology file in Virtuoso Studio design environment

  In OpenAccess, the `DATABASE MICRONS` value is taken from the database units per user unit (`DBUPerUU`) value defined in the technology file. A sample of the OpenAccess technology file is shown below.

  On CDB, the value for `DBUPerUU` for each cellview type can be stored in multiple locations in the technology file, such as the `prop.xx` file, attached technology file, and `.cdsenv` file.

  On OpenAccess 2.2, the database units per user unit (**`DBUPerUU`**) is stored at a single location in the technology file — `viewTypeUnits` subsection of the `controls` section.

  Example

  ```
  controls(
   viewTypeUnits(
   ;( viewType            userUnit       dbuperuu           )
   ;( --------            --------       --------           )

     ( maskLayout          "micron"        2000             )
     ( schematic           "inch"          160              )
     ( schematicSymbol     "inch"          160              )
     ( netlist             "inch"          160              )
   ) ;viewTypeUnits
  ) ;controls
  ```

  After changing the value of `dbuperuu`, recompile and save the technology file by using the Technology File Manager. If you want to use this updated technology file in an open Abstract Generator session, you will need to re-open the library to force the application to re-read the updated technology file (*File – Library – Open*).

© 2023 Cadence Design Systems, Inc.

Related Topics

Exporting LEF in Abstract Generator

Read-Only Technology File in Abstract Generator

© 2023 Cadence Design Systems, Inc.