cadence®

Virtuoso Electromagnetic Solver Assistant 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

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

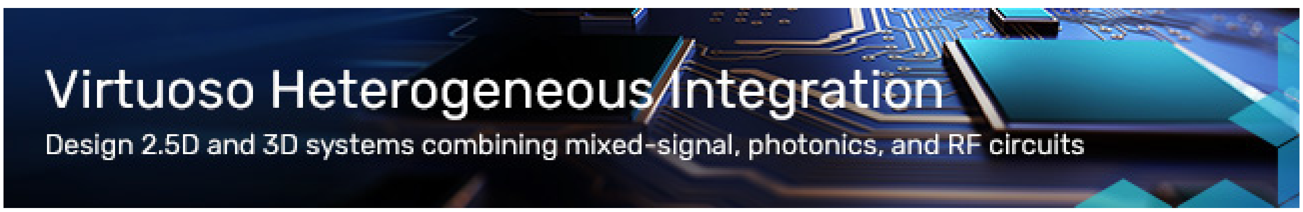
© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

1

Introduction to the Electromagnetic Simulations in Virtuoso RF Solution



After a fully functional layout view of a chip, package or PCB design is ready, it is important to resimulate the design to see the effect of the devices added while preparing the layout. For a Virtuoso RF design, it means running a simulation to study the RF behavior of the passive devices, such as Pcells, inductors, baluns, transformers, MiM/MoM capacitors, and interconnects, that are added to the layout view. Simulators, such as Spectre, require physical model or 3D models to run simulation for such devices. The Electromagnetic Solver assistant in Virtuoso Layout MXL provides an interface to create 3D models for such devices by running the S-Parameter (Scattering Parameters) extraction. These S-Parameters describe the electrical behavior of the linear electrical networks in the design.

For extraction, the Electromagnetic Solver assistant is integrated with two extraction engines: Clarity™ 3D Solver, and EMX® Planar 3D Solver. Depending on your design and the accuracy requirements, you can choose an engine to run extraction and generate 3D models that are saved in touchstone files. Next, you can backannotate these S-parameter 3D models to the schematic view and run Spectre simulations.

For an IC layout, if you have access to the connectivity details, you can choose specific objects in the layout and create an extracted view for those. If you do not have access to nets and connections of devices in the layout, you can run full cellview extraction to create sparam views for the entire layout.

© 2023 Cadence Design Systems, Inc.

Related Topics

Electromagnetic Model Creation in Virtuoso RF Solution

Launching the Electromagnetic Solver Assistant

© 2023 Cadence Design Systems, Inc.

Licensing Requirements

To use the Electromagnetic Solver assistant as part of the Virtuoso RF Solution flow in IC23.1for a package fabric, you require the following licenses:

- Virtuoso Layout Suite MXL

Important

If you open a package layout in Layout MXL, it also requires the Virtuoso RF Option (95560) license.

OR

- Virtuoso Layout Suite EXL + Virtuoso_MultiTech_Framework (95022)

To use the Electromagnetic Solver assistant as part of the Virtuoso RF Solution flow in IC23.1for an IC fabric, you require the following licenses:

- Virtuoso Layout Suite EXL OR Virtuoso Layout Suite MXL

One or more of the following licenses for the solver you want to use:

- SYS300 Clarity 3D Solver, if you want to run high-performance electromagnetic simulation on multi-core computing resources. The adaptive finite element mesh (FEM) refinement technology of Clarity 3D provides consistent accuracy for complicated 3D structures.
- EMX_SOLVER, if you want to use EMX Planar 3D Solver.

Related Topics

Virtuoso Software Licensing and Configuration Guide

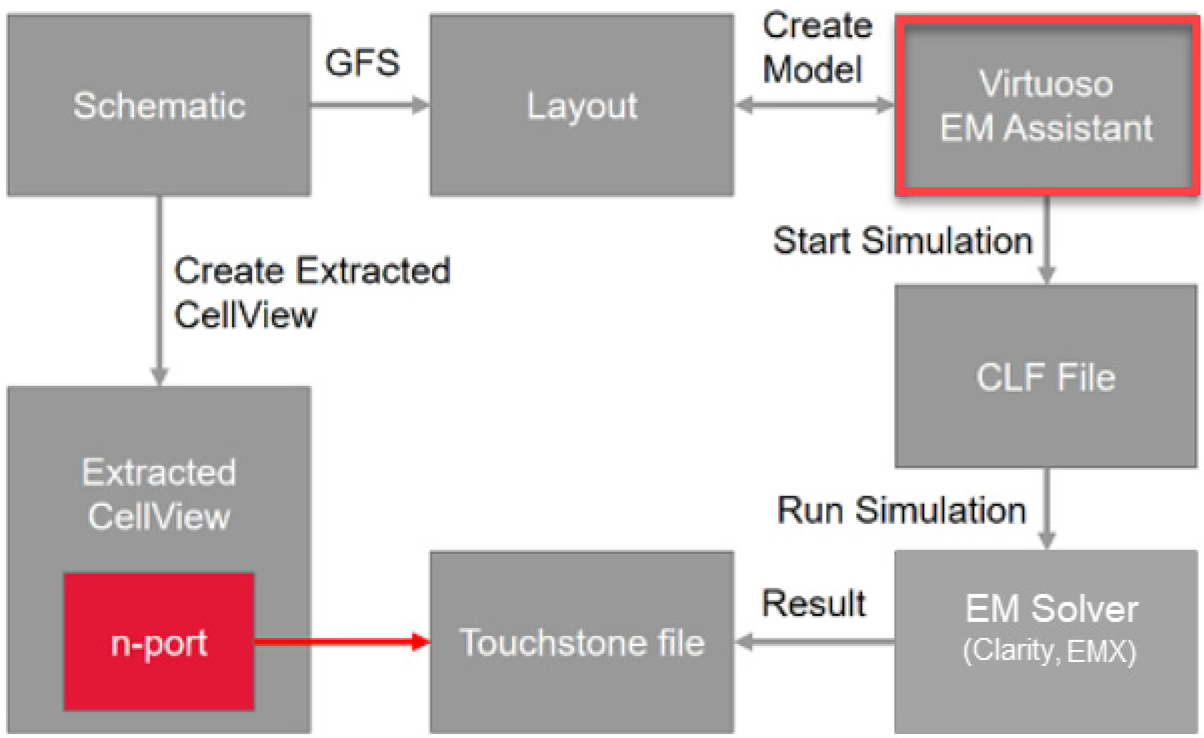
7 Habits of Highly Successful S-Parameters

© 2023 Cadence Design Systems, Inc.

Electromagnetic Model Creation in Virtuoso RF Solution

The Electromagnetic Solver assistant in Virtuoso Layout MXL provides an interface to create 3D models for passive devices in the layout view.

The following flow diagram describes how you can use the Electromagnetic Solver assistant in the Virtuoso RF design flow.



1. After the layout is ready, open it in Virtuoso Layout MXL and launch the Electromagnetic Solver assistant. Next, create a model by selecting the shapes and devices that you need to analyze. The model definition contains all of the information needed to create the physical structure as well as the required settings for simulation by the EM solver.
2. This model is exported to an encrypted data model file (`.clf`) that is automatically imported by the solver for S-Parameter extraction.
3. The solver computes the RF behavior for the shapes identified in the model and saves the output in a touchstone file.
4. You can create an extracted view for use in the ADE Explorer or ADE Assembler setup to run simulations.

© 2023 Cadence Design Systems, Inc.

5. The extracted view contains an n-port that replaces the original net connections for the identified shape or device. This n-port points to the touchstone file in which the RF behavior of the devices is saved. The Spectre simulations read the data from the touchstone file and use that for running simulations.

The integration of the EM engines into the Electromagnetic Solver assistant makes the EM extraction and analysis fully automatic and time saving. You can use a single interface to create models, generate RF data, and save it in a format to be used for running simulations.

Contents of Model Definition File

The model definition in the `.clf` data file includes the following information:

- (Only for an IC layout) The process information that defines layer thicknesses and material properties. This might include one or both of the following:
  - Corner definition from ictTechFile, qrcTechFile, or eadTechFile
  - Custom stackup definition
- (Only for a package layout) The stackup from the technology database
- Interconnects from the layout. These can be a set of nets and instances with an optional rectangle or polygon cutting boundary.
- (Optional) Cutting boundary or die ground size that determines the X-Y extents of the model
- Ports
- Simulation settings

Related Topics

Supported Electromagnetic Solvers

Launching the Electromagnetic Solver Assistant

© 2023 Cadence Design Systems, Inc.

Supported Electromagnetic Solvers

The Electromagnetic Solver assistant supported the following solvers to run extraction and create S-Parameters:

- Clarity 3D Solver: Uses a three-dimensional (3D), full-wave, finite element method (FEM) Clarity 3D Solver that uses the industry-leading parallelization technology to ensure that both meshing and frequency sweeping can be partitioned and parallelized across as many computers, computer configurations, and cores as are available.
- EMX Planar 3D Solver: Uses a planar 3D solver to simulate high-frequency, RF, and mixed-signal integrated circuits. It allows you to accurately and efficiently simulate large RF circuit blocks, characterize the behavior of passive components, and analyze the parasitics due to interconnect.

Important

It is required to set the paths to the solvers you want to use for electromagnetic simulations in the PATH environment variable or the solver-specific environment variables, EMX_PATH or CLARITY_PATH.

You can also use the CLF-specific executables. The first command line argument for the executables should be a `.clf` file. The other arguments should be passed to the actual binary.

```
emx_clf file.clf [list of emx command line options]
gdsview_clf file.clf [list of gdsview command line options]
```

To use the executables, set EMX and GDSview in the $PATH. The executables use the path to a .clf file and invoke EMX/GDSview on it.

Related Topics

License Requirements of Virtuoso RF Solution

© 2023 Cadence Design Systems, Inc.

Launching the Electromagnetic Solver Assistant

To launch the Electromagnetic Solver assistant:

1. Launch Virtuoso.
2. In the Library Manager, select the required library and cell in the *Library* and *Cell* columns.
3. Double-click the name of the layout view.

   A package layout is opened in Layout MXL. An IC layout is opened in the application that was used to save it last. If opened in Layout XL, you can choose *Launch – Layout MXL* to open the layout in Virtuoso Layout MXL.

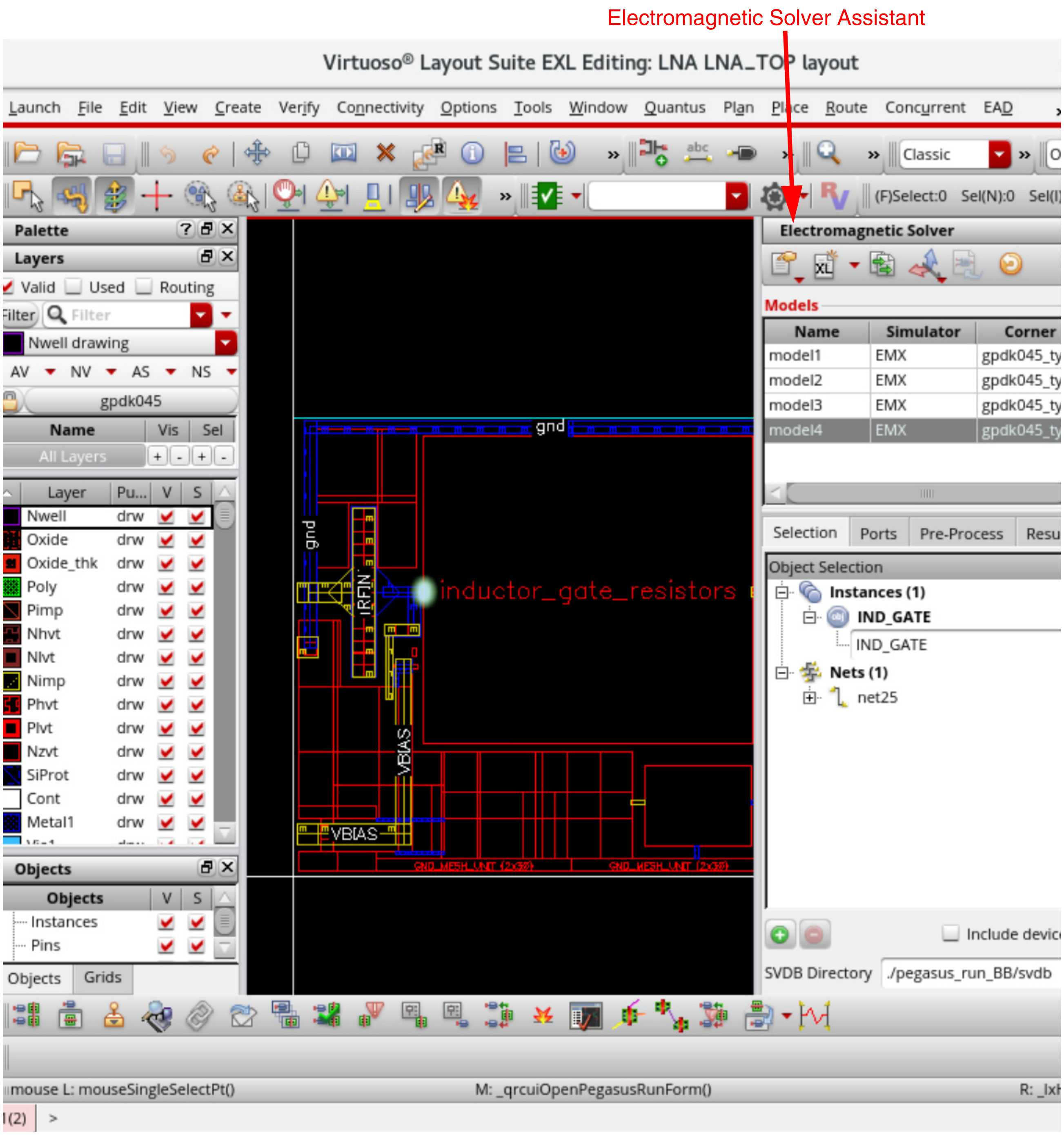
4. Select the `Electromagnetic` workspace from the *Workspace Configuration* drop-down list.

   The `Electromagnetic` workspace makes the following changes in the Layout MXL environment:

   - Displays the Electromagnetic Solver assistant to the right of the layout canvas. At this time, the tool searches for the supported solvers in the paths set by the PATH environment variables. If the solvers are found, details are shown in the Electromagnetic Solver assistant. Otherwise, the tool displays an error in the assistant. In the latter case, you need to close the Virtuoso session and set the PATH environment variable or the solver-specific variables, EMX_PATH and CLARITY_PATH, to add the paths to the solvers.
   - Opens the Navigator assistant and Property Editor assistant
   - Opens the Palette assistant showing the list of layers available in the design
   - Expands the Options toolbar to make the *Dim* icon visible
   - Hides the Floorplanning toolbar

© 2023 Cadence Design Systems, Inc.

The Layout Suite MXL environment window appears as shown below.



© 2023 Cadence Design Systems, Inc.

Important

The above figure shows the Electromagnetic Solver assistant opened for a package layout. This assistant contains some additional fields and tabs when opened for an IC layout.

Related Topics

Licensing Requirements

Virtuoso RF Release Compatibility Matrix

The Electromagnetic Solver Assistant GUI

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

2

Model Creation for Electromagnetic Simulation

The first step towards EM extraction is to define models that contain the instances, nets, or other reference models for which you need the RF data. The Electromagnetic Solver assistant provides various settings that help you create a model, choose a simulator, specify simulator settings, choose objects to be included in model, create ports for the objects, specify criteria for shape simplification, and then extract model definition to be exported to the EM simulator.

Related Topics

Validating Layer Settings for a Package Layout

Validating Ball Grid Array Settings for a Package Layout

Creating a Model

Viewing the Layer Stackup for a Model

Selecting Objects from a Package Layout

Selecting Objects from an IC Layout

Port Creation for Clarity Models in IC Layouts

Specifying Ports for Clarity Models in Package Layout

Specifying Ports for EMX Models in IC Layouts

Specifying Shape Simplification Options for IC Layouts

Protecting Layout Objects from Shape Simplification

Previewing Pre-Processed Layout

© 2023 Cadence Design Systems, Inc.

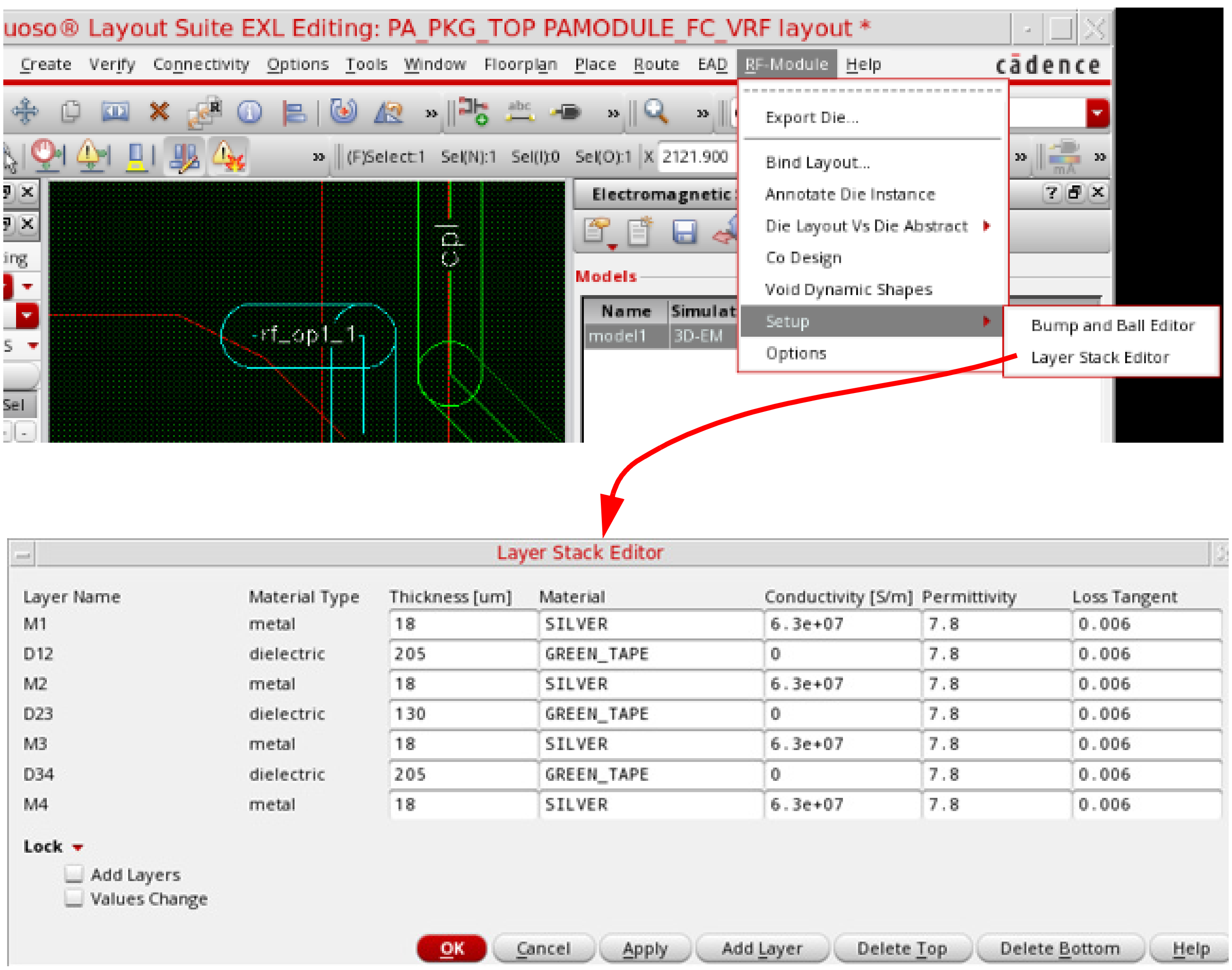
Validating Layer Settings for a Package Layout

The layer thicknesses, metal conductivity, dielectric permittivity (Dk) and dielectric loss tangent (Df) in the package layout must match the fab parameters. This ensures that the captured electrical response is correct.

These details are provided in the technology file. You can also view these in the Layer Stack Editor and make changes, if required.

To validate the layer settings for a package design:

1. Choose *RF-Module – Setup – Layer Stack Editor* to open the Layer Stack Editor.



   Validate the details in the form.

2. If the dielectric properties for the metal layers is missing, fill in the data.

© 2023 Cadence Design Systems, Inc.

3. (Optional) Add a dielectric layer on top or bottom of your design layer.
4. (Optional) If the layer that exists on top of your design layer is not required, click *Delete Top* to delete that top layer. Next, click *Add Layer* to add a new metal or dielectric layer by using the Add Layer form.

Related Topics

Add Layer Form

Layer Stack Editor form

© 2023 Cadence Design Systems, Inc.

Validating Ball Grid Array Settings for a Package Layout

For a physical design, you need to validate the bump and ball parameters of the required instances to ensure that the layout can be:

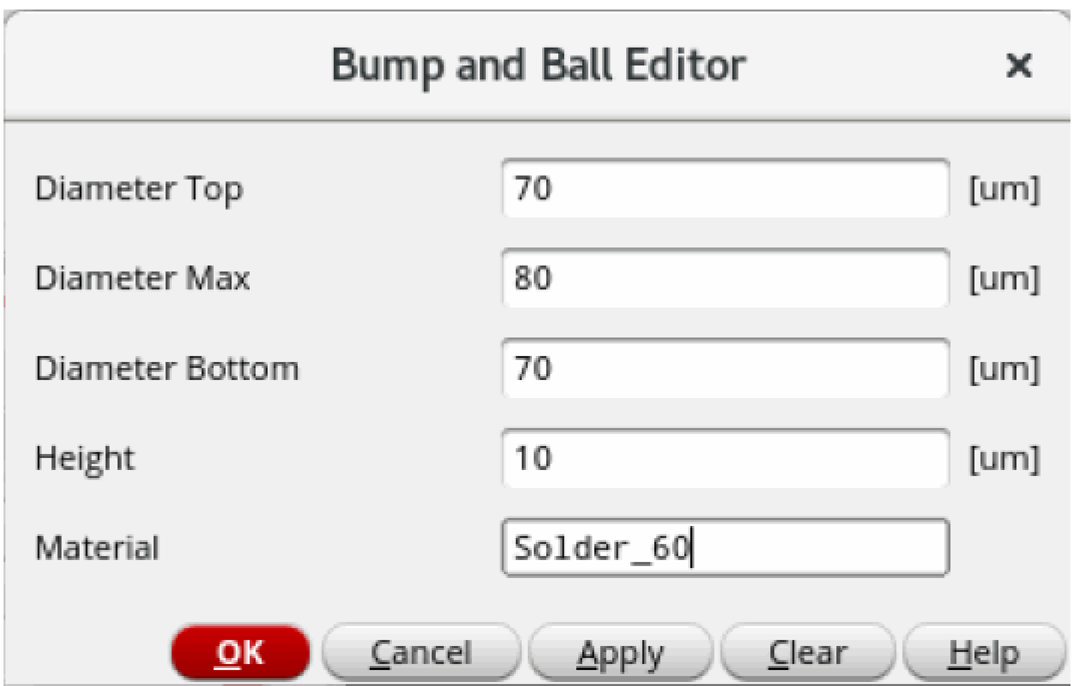
- exported to Allegro.
- used by EM for EM calculation.
- used by 3D Viewer to display bumps and balls in 3D.

If the bump and ball parameters were not set when the layout was imported from Allegro, the Bump and Ball Editor form sets these parameters.

To check the bump and ball dimensions:

1. Select an instance, such as a flip chip die, a ball grid array (BGA) instance, or a surface-mount device (SMD) instance on the package layout.
2. Choose *Module – Setup – Bump and Ball Editor*.

   The Bump and Ball Editor form is displayed.



3. Enter the physical dimensions of the solder ball.

   Ensure that the bump height is greater than the thickness of the solder mask specified on the Layer Stack Editor form.

© 2023 Cadence Design Systems, Inc.

4. Click *OK*.
5. The specified parameters are applied to the bumps and balls of the selected instance.

Related Topic

Bump and Ball Editor Form

© 2023 Cadence Design Systems, Inc.

Configuring Process Settings for IC Layouts

For an IC layout, you also need to define the process settings that provide the physical layer stack information required by EM solvers to create and solve a model.

For the models that use Clarity 3D Solver, layer stack information is available in one of the following files provided by the foundry:

- ICT technology file
- QRC technology file

If you do not have these files, you can provide your own process corner `.emproc` files with custom stackup information. In a process corner file, you need to add references to the technology files or add custom stackup details.

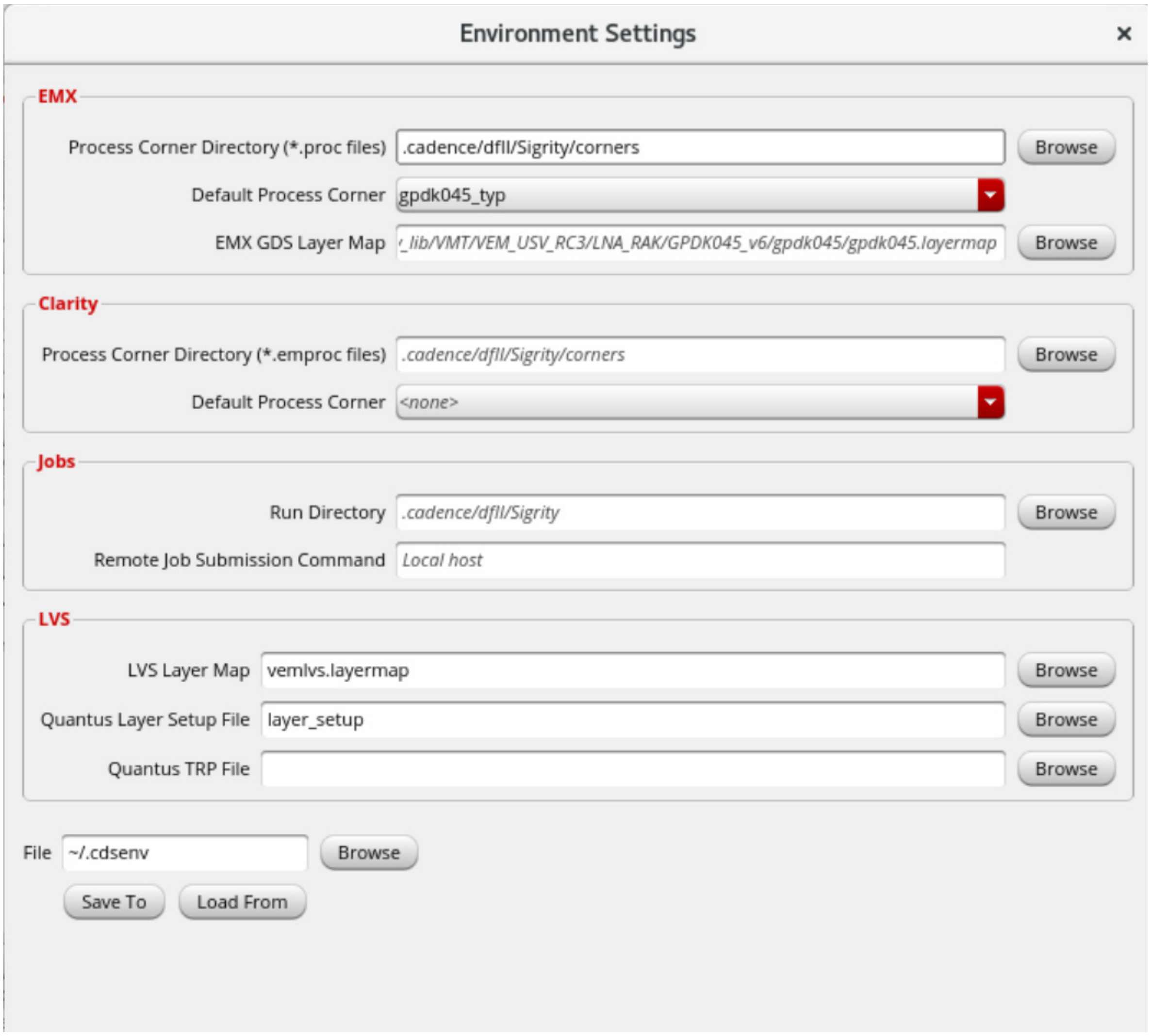
For the models that use EMX 3D planar solver, you just need a standard `.proc` EMX process file. Check with your foundry if they provide this file. Check the *EMX User Guide* for more information on the EMX process file format.

When you have process corner files or technology files for your design, configure the process corner settings for the EM models by performing the following steps:

1. Choose *Settings – Environment* on the toolbar of the Electromagnetic Solver assistant.

   The Environment Settings form is displayed.

© 2023 Cadence Design Systems, Inc.



2. For the models that use EMX 3D Planar, specify the path to the process corner directory and the name of the default process corner in the *EMX* section. Ensure that the `.proc` files are saved at this path.

   The *Process Corner Directory* field in the *EMX* section depends on the value set for the `emxProcessCornerDirectory` environment variable. When this variable is set to `""`, the tool considers `.cadence/dfII/Sigrity/corners` as the path. Similarly, the *Default process corner* field in the *EMX* section shows the corner name specified by the `emxProcessDefaultCorner` environment variable. When multiple process corners are defined, the tool uses the corner specified in this field.

   Note: If the default process corner is not found in the process corner directory, the tool uses the first found process corner in the alphabetical order.

© 2023 Cadence Design Systems, Inc.

3. For the models that use Clarity 3D Solver, specify the path to the process corner directory and the name of the default process corner in the *Clarity* section.
4. By default, the *Process Corner Directory* field in the *Clarity* section depends on the value set for the `processCornerDirectory` environment variable. When this variable is set to `""`, the tool considers `.cadence/dfII/Sigrity/corners` as the path. Similarly, the *Default process corner* field in the *Clarity* section shows the corner name specified by the `processDefaultCorner` environment variable. When multiple process corners are defined, the tool uses the corner specified in this field.

   Note: If the default process corner is not found in the process corner directory, the tool uses the first found process corner in the alphabetical order.
5. (Optional) If the process corner settings are saved in a `.cdsenv` file, click *Load From* to load the settings from that file.

Related Topics

Environment Settings Form

Format of the Process Corner Setting File Referring to an ICT or QRC Technology File

Format of the Custom Process Corner Setting File

emxProcessCornerDirectory

emxProcessDefaultCorner

processCornerDirectory

processDefaultCorner

© 2023 Cadence Design Systems, Inc.

Format of the Process Corner Setting File Referring to an ICT or QRC Technology File

The standard syntax of the `.emproc` file that refers to an external ICT or QRC technology file for process corner information is shown below.

```
(externalStackup
   (filePath t_path)              ; path to ICT/qrcTechFile
   (layerMap
      l_mappingICTtoVirtuoso      ; list of (t_ictName t_virtuosoName)
      )
   (metalLayers
     ; List of metal layers embedded inside substrate
      (t_name elevation f_elevation thickness f_thickness
      {conductivity  f_conductivity | sheetResistance f_sheetResistance })
   )
   (excludeLayers l_ictlayerNames )   ; list of ICT layers not to be exported
)
(layerElevationAdjustment
   (t_layerName f_heightAdjustment)   ; adjusts height of the given layer in nm
)
(dielectricSimplification
   {"none" | "weightedAverage" | "singleDielectric" permittivity f_permittivity
lossTangent f_lossTangent}
)
(substrate
     ; List of dielectrics top-down
   (t_name thickness f_thickness permittivity f_permittivity
   {conductivity   f_conductivity | resistivity   f_resistivity })
      ...
)

(includeLPPs
   l_includeLPPs
   ; list of (t_layer t_purpose) LPPs, by default only drawing purpose is included
)

(outercoating
   (tsv_via_name thickness f_thickness permittivity f_permittivity )
   ; tsv_via_name is one of the vias in the external stackup
)

(materialFile
   (filePath t_filePath )
   ; Corner frequencies for D-Sarkar model
   (enforceCausality lowFreqCorner f_lowFreqVal highFreqCorner f_highFreqVal
    baseFreq f_baseFreqVal )

   (materials
   ; Materials for layers and dielectric
   (t_ICTLayer t_conductorMaterial t_dielectricMaterial )
   )
   (topMaterial t_topMaterialName )  ; material for dielectric above top metal
   (bottomMaterial t_bottomMaterialName )  ; material for dielectric below the
bottom metal
)
```


© 2023 Cadence Design Systems, Inc.

Tags in the EM Process Corner Settings File

This file contains the following tags:

- `externalStackup`: Indicates that the information about the process corner needs to be taken from an external ICT or QRC file provided at the given file path.

  The `externalStackup` tag contains the following tags:

  - `layerMap`: Specifies the mapping of ICT layers to the layers in your layout.

    You can specify more than one Virtuoso layers in a `layerMap` tag. The first argument is the ICT layer name. All the other arguments are names of the Virtuoso layers.

    ```
    ("ICT_layer" "layer1" "layer2" "layer3" …)
    ```

    If the name of a layer is same in Virtuoso and in the ICT file, it is automatically mapped and no explicit layer mapping is required. To override auto-mapping for any layer you do not want to map, map it to `nil`. For example, for example,

    ```
    ("Metal1" nil)
    ```

    In this case, `Metal1` is not exported to the solver.

  - `metalLayers`: Specifies a list of metal layers embedded in the substrate. Use this tag to provide details of the metal layers that are inside the substrate but are not referenced in the external ICT or QRC file. The elevation of these layers is measured from the top of the substrate layer to the top of the metal layer below it. Therefore, it must be a negative value less than the substrate thickness.

  - `excludeLayers`: Specifies a list of layers to be excluded. Any layer to be excluded does not get exported to the CLF file that is sent as an input to the solver. The names specified in this list must be the layer names from the ICT file, not the layers in the layout.

    Though you can exclude any layer using this tag, it is particularly useful when you want to exclude a specific layer from a MIMCAP. For example, you want to send only the metal layers of a MIMCAP to the solver and exclude the capacitor and the vias connecting the capacitor to the metal layers.

    Note: If you are excluding layers from a model even though the selected objects are embedded on those layers, you would also need to later exclude those objects from extracted views so that the objects present in the schematic view are used. For details on how to exclude specific instances from the extracted view, refer to Creating Extracted Views from Models.

© 2023 Cadence Design Systems, Inc.

- `layerElevationAdjustment`: Contains a list of layer name–height adjustment value pairs that you can use to shift the MIMCAP metal layers so that the desired capacitance value for the cell is maintained. Each pair specifies the name the MIMCAP metal layer you need to shift and the value (in nanometers) by which you need to adjust the height of the MIMCAP metal layer. A positive value shifts the layer up, and a negative value shifts it down. The shift helps in emulating the effect of the high-K dielectric used between the MIMCAP plates.
- `substrate`: Contains a list of substrate layers. The topmost layer is listed first, followed by other layers in the same sequence in which they exist in the design.

  For each substrate, you can specify the following:

  - `name`: Name of the substrate layer
  - Thickness (in µm) of an additional dielectric to be inserted above the bulk substrate and below the first layer specified in the ICT file
  - Permittivity value (in F/m) of the dielectric to be inserted above the bulk substrate and below the first layer specified in the ICT file
  - Conductivity (in S/m) of the dielectric to be inserted above the bulk substrate and below the first layer specified in the ICT file
  - Resistivity (in Ω*cm) of the dielectric to be inserted

  Conductivity and resistivity are mutually exclusive settings. You can specify either one of these.

  Note: There is no limit for the number of substrate layers in a stackup, but it is mandatory to specify at least one.
- `includeLPPs`: Contains a list of layer-purpose pairs (LPPs) from which the shapes are to be exported to the `.clf` extracted model file. If not specified, only the shapes from the default drawing purpose (* drawing) are extracted.
- `dielectricSimplification`: Specifies how to set the permittivity and loss tangent values for the simplified dielectric layers that are created by combining all dielectric layers between two metal layers into one. This tag supports these values:

  - `"weightedAverage"`: Calculates the average of permittivity and loss tangent for all dielectric layers combined into one.
  - `"singleDielectric"`: Uses the values specified by the `permittivity` and `lossTangent` properties of this tag.

  Note: If the stackup information used by Clarity has `dielectricSimplification` set to `"none"`, it is considered as `"weightedAverage"`.

© 2023 Cadence Design Systems, Inc.

- `outerCoating`: Specifies the thickness and permittivity of the outer coating to be used to insulate a layer through silicon via.

- `materialFile`: Specifies the path to a `.cmx` material file to be used by the simulator. A material file contains information about the various properties of the different materials used in your design. For example, a material file contains the conductivity or sheet resistance of a material at different temperature and frequencies.. In addition, it provides the names of materials to be used for different ICT layers and dielectric layers.

  The `materialFile` tag contains the following tags:

  - `filePath`: Specifies the path to the `.cmx` file from where the simulation environment can load information about the different materials used in your design. For example, Clarity 3D Workbench loads the material information in the Material Manager form.

  - `enforceCausality`: Enforces causality in material properties. When this feature is enabled, all dielectric materials are fitted with D-Sarkar Model. In this tag, you can also specify the low frequency, high frequency and base frequency values to be used by the D-Sarkar model in Clarity 3D Workbench. These values are also visible in the Material Manager form.

  - `materials`: Specifies the metal names for the ICT layer, conductor material, and dielectric material. A conductor can either have only a conductor material, or a fill-in material, or both.

  - `topMaterial`: Specifies the name of the material to be used for the dielectric above the top metal.

  - `bottomMaterial`: Specifies the name of the material to be used for the dielectric below the bottom metal.

    For details on how to view and edit material information in the Material Manager form and the D-Sarkar model, refer to *Managing Material Information* in Clarity 3D Workbench User Guide.

An example of a `.emproc` file that includes the external stackup from an ICT file follows.

```
externalStackup
   (filePath "./gpdk180_typ.ict")
   (layerMap
      ("Metal6" nil)           ; ; Example of how to override automapping in case
        ; the layer is called Metal6 in Virtuoso and ICT, but should not be exported
        ;to the solver

      ("Via2MIM" "Via2")       ; Via2MIM to be mapped
   )
   (metalLayers
      ("metalX" elevation -20 thickness 1 sheetResistance 0.05)
   )
```


© 2023 Cadence Design Systems, Inc.

```
   (excludeLayers "MetalA" "MetalC" )
)
(substrate
   ("Si" thickness 300 permittivity 11.9 resistivity 2)
   ("Si2" thickness 200 permittivity 11.8 conductivity 11)
)

(outercoating
   ("TSV" thickness .4 permittivity 4.9 ) ; "TSV" is one of the vias in the external
   ; stackup
)
(materialFile
   ( filePath "./myLib/test.cmx" )

   (enforceCausality lowFreqCorner 0.02 highFreqCorner 2e6 baseFreq 2000)

   (materials
      ( "Metal3" "Copper" "dielec" )
      ( "Metal4" "Copper" )
   )

   (topMaterial "MyDiel2" )
   (bottomMaterial "MyDiel3" )

)
```

Related Topics

Configuring Process Settings for IC Layouts

© 2023 Cadence Design Systems, Inc.

Format of the Custom Process Corner Setting File

The standard syntax used to define a custom stackup in a `.emproc` file is shown below.

```
(customStackup
   (metalLayers
     ; Listof metal layers in top-down order
      (t_name elevation f_elevation thickness f_thickness
      {conductivity  f_conductivity | sheetResistance f_sheetResistance })
   )
   (dielectricLayers
      ; List of dielectric layers above the substrate in top-down order
      (t_name thickness f_thickness permittivity f_permittivity lossTangent
       f_lossTangent)
    )
   (dielectricLayersBackside
      ; List of dielectric layers below the substrate in top-down order
      (t_name thickness f_thickness permittivity f_permittivity lossTangent
      f_lossTangent)
    )

   (viaLayers
     ; List of vias in top-down order
      (t_name topMetal t_topMetal bottomMetal t_bottomMetal conductivity
      f_conductivity)
   )
   (designScale t_designScale )   ; default is 1.0, less than 1.0 is shrink
)
   (layerElevationAdjustment
   (t_layerName f_heightAdjustment)   ; adjusts height of the given layer in nm
   )
   (dielectricSimplification
   {"none" | "weightedAverage" | "singleDielectric" permittivity f_permittivity
   lossTangent f_lossTangent}
   )
   (substrate
     ; List of dielectrics top-down
   (t_name thickness f_thickness permittivity f_permittivity
      { conductivity f_conductivity | resistivity f_resistivity } )
      ...
   )
   (outercoating
   (tsv_via_name thickness f_thickness permittivity f_permittivity )
   ; tsv_via_name is one of the vias in the external stackup
   )

   (includeLPPs
   l_includeLPPs
   ; list of (t_layer t_purpose) LPPs, by default only drawing purpose is included
   )

(materialFile
   (filePath t_filePath )
   ; Corner frequencies for D-Sarkar model
   (enforceCausality lowFreqCorner f_lowFreqVal highFreqCorner f_highFreqVal
    baseFreq f_baseFreqVal )

   (materials
```


© 2023 Cadence Design Systems, Inc.

```
    ; Materials for layers and dielectric
    (t_ICTLayer t_conductorMaterial t_dielectricMaterial )
    )
    (topMaterial t_topMaterialName )  ; material for dielectric above top metal
    (bottomMaterial t_bottomMaterialName )  ; material for dielectric below the
     bottom metal
)
```

Tags in the Custom Process Corner Settings File

This file can contain the following tags:

- `customStackup`: Contains the following subtags:
  - `metalLayers`: Specifies a list of metal layers in the design. For each metal layer, you can specify the name, elevation, thickness, permittivity, and loss tangent.

    For metal layers below the substrate, specify a negative value for the elevation argument. In this case, the elevation is measured from the top of the substrate to the top of the metal layer below the substrate.
  - `dielectricLayers`: Specifies a list of dielectric layers above the substrate. For each dielectric layer, you can specify the name, thickness, permittivity, and loss tangent.
  - `dielectricLayersBackside`: Specifies a list of dielectric layers below the substrate. For each dielectric layer, you can specify the name, thickness, permittivity, and loss tangent.
  - `viaLayers`: Specifies the details of vias or through silicon vias. When you have a through silicon via or an interposer IC, you also need to specify the `outerCoating` tag.
  - `designScale`: Specifies a scaling factor for the layout. The scaling factor is used when the size of the drawn layout is not the same as the size of the manufactured layout.
- `layerElevationAdjustment`
- `dielectricSimplification`
- `substrate`
- `includeLPPs`
- `outerCoating`
- `materialFile`

© 2023 Cadence Design Systems, Inc.

An example of a `.emproc` file that defines a custom stackup is given below.

```
(customStackup
 (metalLayers
  ("AP" elevation 6 thickness 1 conductivity 6e6)
  ("M4" elevation 5 thickness 0.5 conductivity 6e6)
  ("M3" elevation 4 thickness 0.5 conductivity 6e6)
  ("M2" elevation 3 thickness 0.5 conductivity 6e6)
  ("M1" elevation 2 thickness 0.5 conductivity 6e6)
  ("MB" elevation -100.8 thickness 2.0 conductivity 6e6)
 )

 (dielectricLayers
  ("D1" thickness 10 permittivity 3.9 lossTangent 0)
 )

 (dielectricLayersBackside
  ("BD1" thickness 3 permittivity 6.70000 lossTangent 0.00000)
 )

 (viaLayers
  ("RV" topMetal "AP" bottomMetal "M4" conductivity 6e6)
  ("VIA3" topMetal "M4" bottomMetal "M3" conductivity 6e6)
  ("VIA2" topMetal "M3" bottomMetal "M2" conductivity 6e6)
  ("VIA1" topMetal "M2" bottomMetal "M1" conductivity 6e6)
  ("TSV" topMetal "M1" bottomMetal "MB" conductivity  6e6)
 )
)

(outerCoating
    ("TSV" thickness .4 permittivity 3.9)
)

(substrate
    ("Si" thickness 100 permittivity 11.9 conductivity 10)
)
```

Related Topics

Configuring Process Settings for IC Layouts

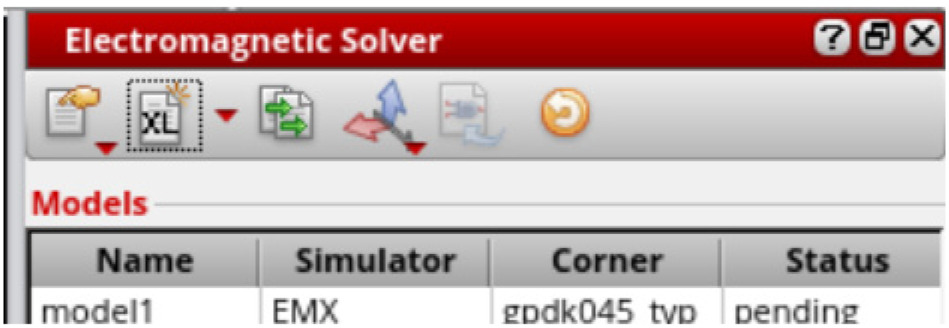
Format of the Process Corner Setting File Referring to an ICT or QRC Technology File

© 2023 Cadence Design Systems, Inc.

Creating a Model

To create a new model for an electromagnetic simulation:

1. Open the IC or package layout in Virtuoso Layout MXL.
2. Press the `Shift+F` bindkey to make all the layers in the layout hierarchy visible.
3. Press the `F` key to fit the complete layout in the canvas.
4. In the Electromagnetic Solver assistant, click  on the toolbar of the Electromagnetic Solver assistant.



A new model is added in the *Models* table.

Observe the following:

- By default, the models are named as `modelx`, where `x` is an incremental number. However, you can double-click in the model name field and specify another name.
- The default simulator assigned to a model for a package design is `EMX`. The default simulator for a package design is `Clarity`.

  For IC layouts, you can also choose to use `EMX`.

  You can specify the default simulators to be used for package and IC designs by using the `vem.ic defaultSimulator` and `vem.package defaultSimulator` environment variables, respectively.
- The Status column of a model shows its simulation run status. By default, it is set as *Pending*.
- For IC layouts, the *Models* table shows an additional column named *Corner*. In this column, you can choose the name of the process corner to be used for your model. You can use the drop-down list to choose a corner name from the list of available corners. EM Assistant populates the list of corner names by reading the process corner definition files (`.emproc` or `.proc` files) provided in the directory specified in the Environment Settings form.

© 2023 Cadence Design Systems, Inc.

Related Topics

Configuring Process Settings for IC Layouts

Environment Settings Form

defaultSimulator (vem.ic)

defaultSimulator (vem.package)

© 2023 Cadence Design Systems, Inc.

Creating a Copy of a Model

You can create a copy of a model and reuse it to create another model.

To create a copy of a model,

➡ Right-click a model name and choose *Duplicate*.

A new row is added to the *Models* table. The new model is named as `old-model-name_copyx`, where *x* is an incremental number. Settings from all the tabs of the Electromagnetic Solver assistant are copied from the original to the duplicate model.

© 2023 Cadence Design Systems, Inc.

Deleting a Model

To delete a model, right-click anywhere in the row of the model and choose *Delete*. You can delete multiple models from EM Assistant simultaneously.

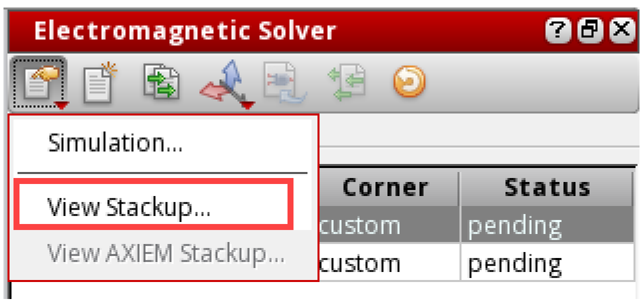
© 2023 Cadence Design Systems, Inc.

Viewing the Layer Stackup for a Model

Each model uses the layer stackup information provided for the process corner that it is using.

To review the layer stackup:

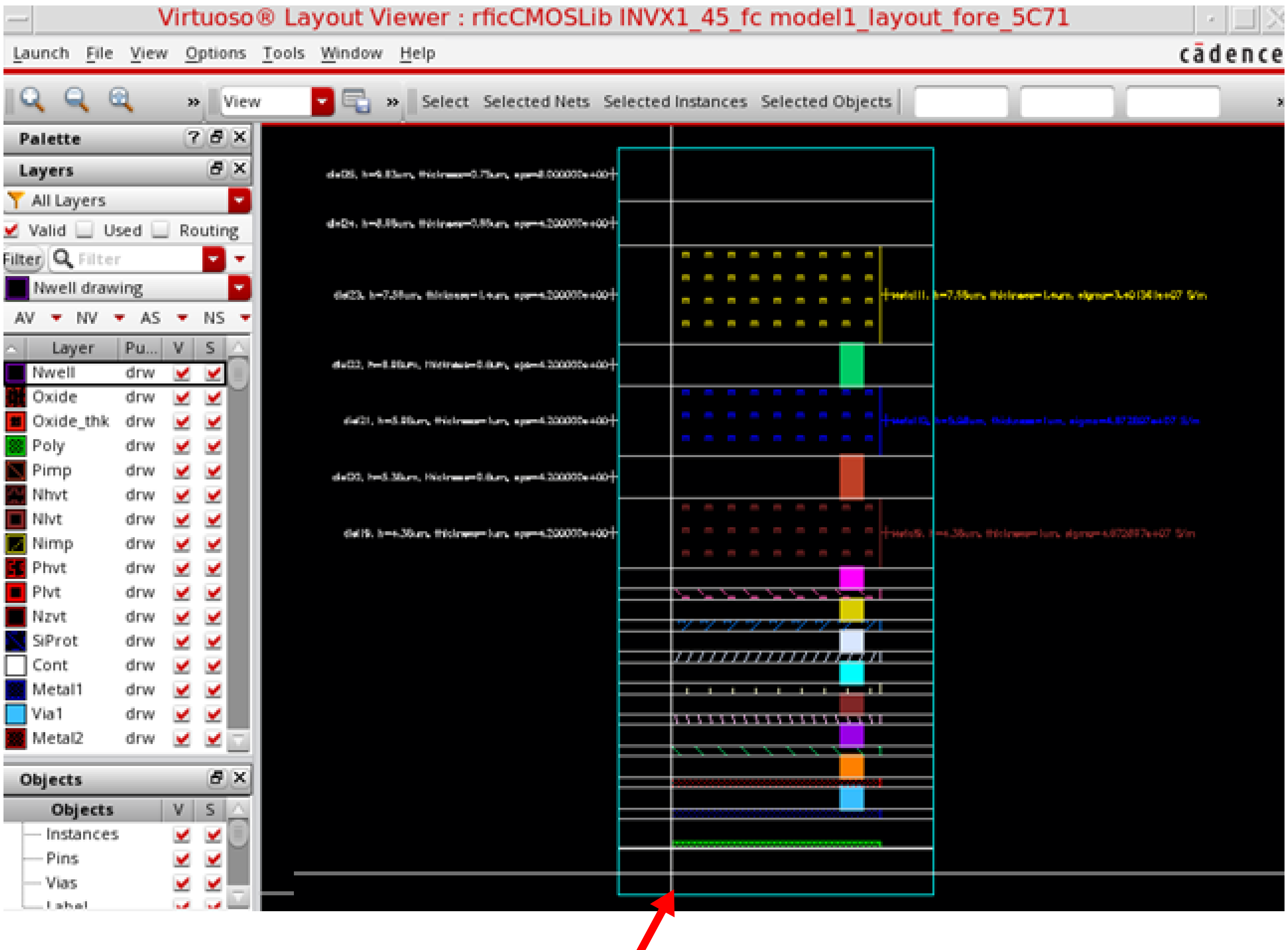
1. Select the row for the model.
2. Choose the *View Stackup* command from the toolbar of the Electromagnetic Solver assistant.



The Virtuoso Layout Viewer window is displayed. It shows the layer stackup as defined in the `.emproc` or `.proc` file associated with the model.

© 2023 Cadence Design Systems, Inc.

The following figure shows an example of the layer stackup:



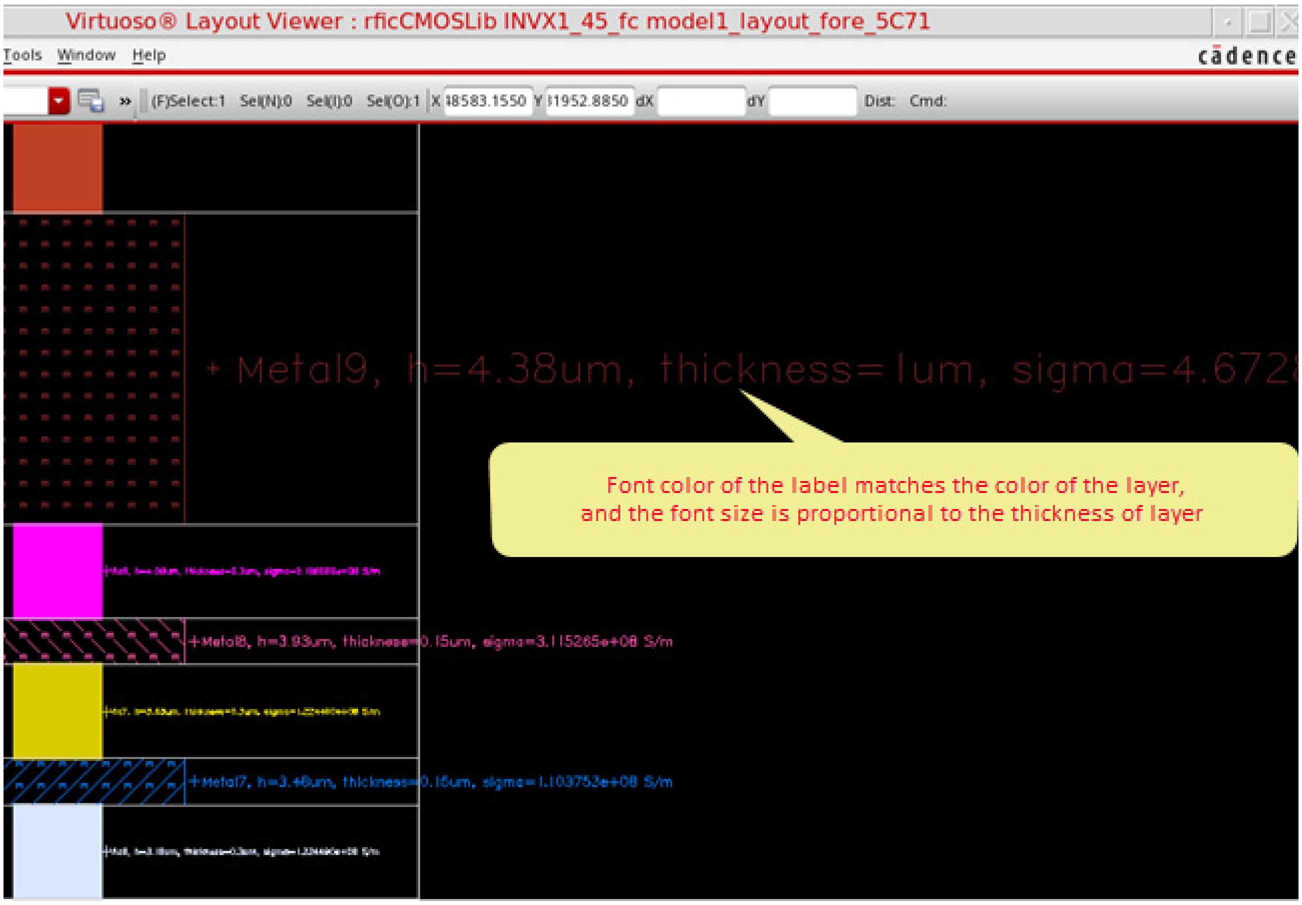
Substrate layer

The substrate layer is shown at the bottom, and the dielectric and metal layers are stacked above that. If the thickness of the substrate specified in the `.emproc` file is greater than 10 µm, it is scaled down.

The labels to the left of the stack show the details of dielectric layers, whereas the labels to the right of the stack show the details of metal layers and vias. Each layer has a supporting label that provides information about it, including the layer name, height, thickness, and permittivity. The font color of the label matches the color of the layer, and the font size is

© 2023 Cadence Design Systems, Inc.

proportional to the thickness of layer. If a label is not visible for any layer, you would need to zoom in to read it.



Viewing the Effective Layer Stackup

Depending on the geometry of the design, the layer stackup might get processed or simplified before running simulations, therefore, it is helpful to view the effective stackup that is used by the simulator.

For models with Clarity simulator, you can view the effective stackup that will be used for simulation run, open the simulator interface, and view the stackup. Clarity automatically adjusts the thickness of the top dielectric to match it to the bump height. Therefore, you might see a difference in the layer stackup displayed in the Sigrity environment.

For models with EMX simulator, you can use the commands on the *Pre-Process* tab of the assistant to process the layer stackup.

© 2023 Cadence Design Systems, Inc.

To view the effective layer stackup for EMX models, you can choose *View EMX Scaled Stackup* or *View EMX Unscaled Stackup* from the toolbar of the assistant

Related Topics

VEM_PDF_VIEWER

© 2023 Cadence Design Systems, Inc.

Selecting Objects from a Package Layout

While selecting objects from a package layout, you can select instances or nets from the layout canvas or the Navigator assistant. In addition, for a cross-fabric design, you can select a reference model to be included in the currently selected model.

If you are interested in considering only a section of a selected object, you can use a cutting boundary to define that section of the layout.

If you have a CMX material file to be used by the models with simulator set as `Clarity`, create a config file and specify the path to that file.

To select the objects to be included in a model:

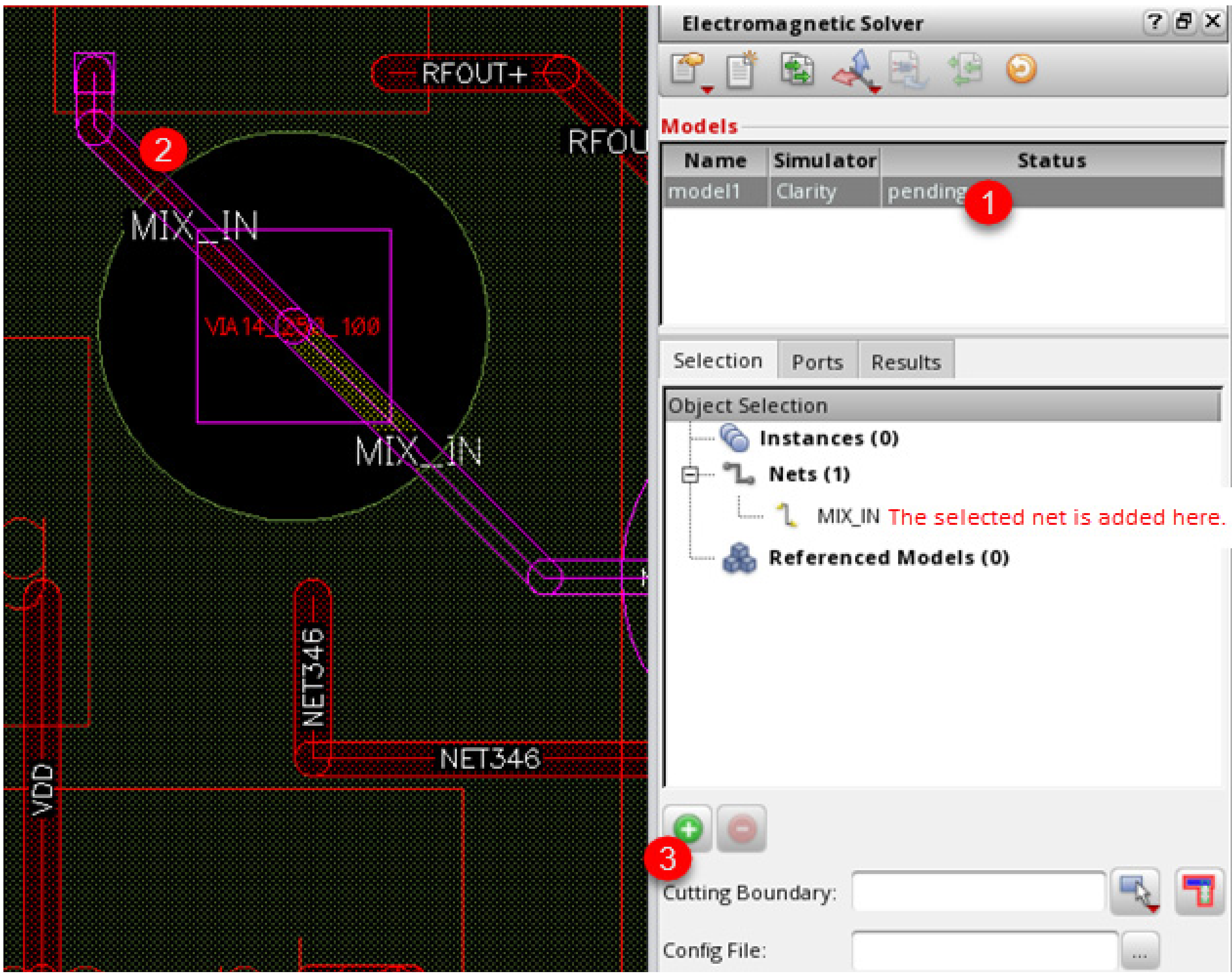
1. Select a model in the *Models* table.
2. On the layout canvas, select an instance or net that you want to add to the model.

   Alternatively, select these in the Navigator assistant.

   Note: When you add a net in a model, ensure that all the net connections are to the top-level schematic pins or to schematic instance pins. You cannot add a net that ends at a pin that is not at the top level of the schematic.
3. Click *Add selected instances/nets* ( ) on the *Selection* tab.

© 2023 Cadence Design Systems, Inc.

The selected net or instance is added to the *Object Selection* list. In the example of a package design shown below, net `MIX_IN` is selected and added to the assistant.

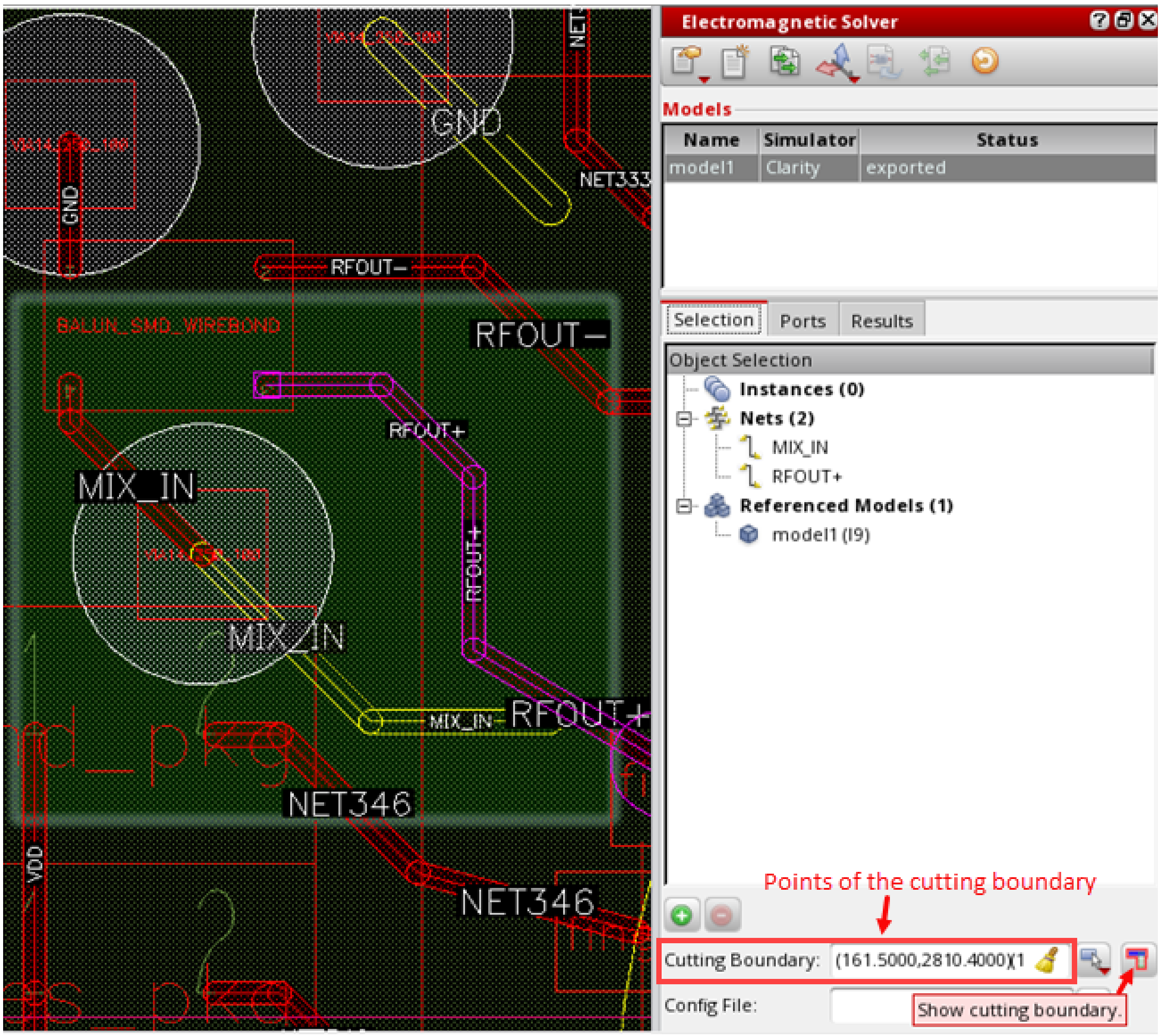


4. Similarly, select more nets or instances to be placed in the same model.

5. (Optional) To create a cutting boundary (rectangular or polygonal) around the selected objects, choose a shape from the *Select cut type* drop-down command list and draw a boundary. This defines the boundary for an interconnect to be included in the model. The interconnects connecting the selected nets and instances from all layers are included in the model.

© 2023 Cadence Design Systems, Inc.

In the example shown below, the model includes only those shapes connected to `MIX_IN` that are within the defined cutting boundary. Any shape connected to this net outside of this boundary is not included in the model.



The rectangle or polygon is highlighted on the layout. The details of the points defining this boundary are also added to the *Cutting Boundary* field

6. (Optional) Click *Show Cutting Boundary* to toggle the visibility of the cutting boundary for the model.

© 2023 Cadence Design Systems, Inc.

7. (Optional) To connect a CMX material file to be used by Clarity 3D Solver while running simulation for a package layout, specify the path to that config file in the *Config File* field on the *Selection* tab.

   Note: If you select a BGA, bond wire, or die for inclusion in the model, it is not directly added to the model. Instead, the nets connected to it are added. During port generation, coaxial ports are created at the points the net connects to the BGA, bond wire, or die. Similarly, for SMD instances, vertical ports are created where the nets are connected to that instance.

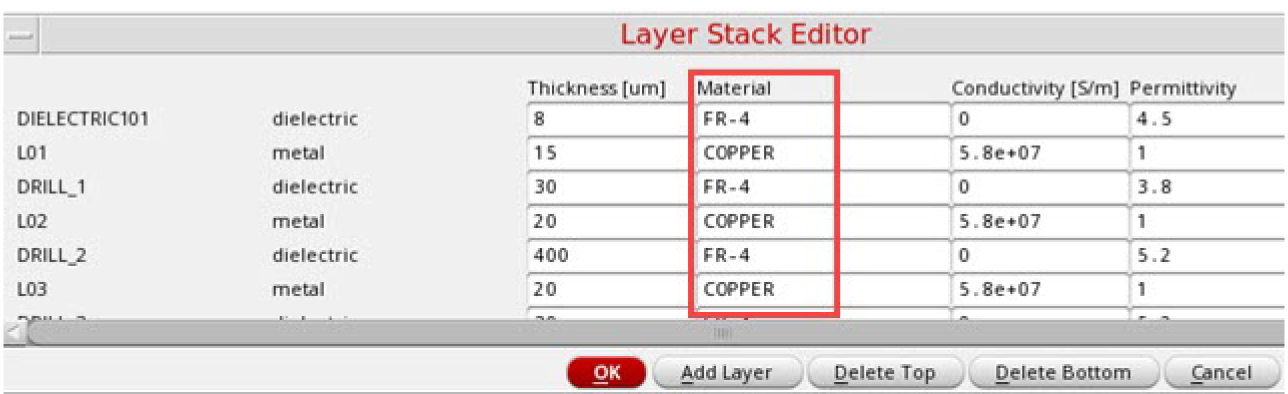
Format of a Config File for a Package Layout

While selecting objects for a model, you can provide information for the CMX material file through a config file. The config file must be saved with the `.emconf` extension.

The format of the config file is shown below.

```
(emConfig
    (materialsFile
       (filePath "./fileName.cmx")
       (enforceCausality enforceCausality lowFreqCorner f_lowFreqVal
        highFreqCorner f_highFreqVal baseFreq f_baseFreqVal )
    )
)
```

The CMX file contains the details for the materials specified for one or more layers in the Layer Stack Editor form.



When you run a simulation for the model, the tool exports the specified CMX file to the EM simulator. The simulator uses the conductivity and permittivity details for the specified layer from the CMX file. For other layers, it uses the conductivity or permittivity details specified in the Layer Stack Editor form.

© 2023 Cadence Design Systems, Inc.

Related Topics

Selecting Objects from an IC Layout

Highlighting Selected Objects on the Layout Canvas

Validating a Flip-Chip Die in a Package Layout

Removing Selected Objects

Validating a Flip-Chip Die in a Package Layout

© 2023 Cadence Design Systems, Inc.

Selecting Objects from an IC Layout

To select the objects to be included in a model:

1. Select a model in the *Models* table.
2. On the layout canvas, select an instance or net that you want to add to the model.

   Alternatively, select these in the Navigator assistant.
3. (Optional) In case of a hierarchical design, traverse the design.

   To descend into an instance, do one of the following:

   - ❑ Expand an instance in the *Navigator* assistant to view all the instances it contains.
   - ❑ Right-click an instance on the layout canvas and choose *Edit in Place* or *Descend Edit*.

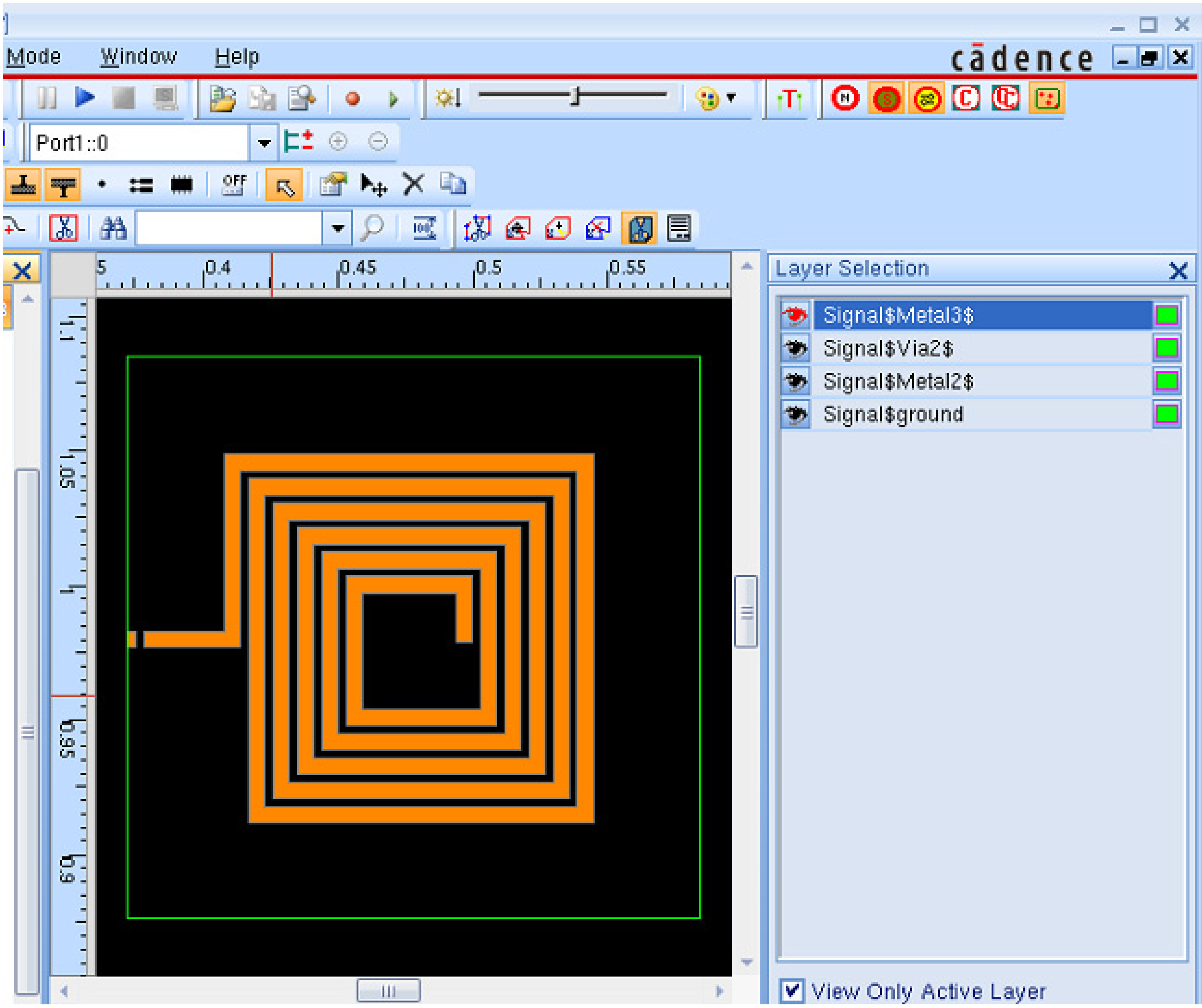
   Select nets or instances from any level.

   The name of a selected object shows the complete hierarchical path to the selected instance or net where the name of each level is separated by a `/`.
4. Click *Add selected instances/nets* () on the *Selection* tab.
5. Similarly, select more nets or instances to be placed in the same model.
6. (Optional) Create a cutting boundary (rectangular or polygonal) around the selected objects. This defines the boundary for an interconnect to be included in the model. The interconnects connecting the selected nets and instances from all layers are included in the model.
7. In the *Current Return Path* field, choose one of the following current return path types:

   - ❑ *Local Reference*: You need to find the correct ground for each port by connecting it to the real ground metal or the plane below it or next to it.

© 2023 Cadence Design Systems, Inc.

- *PEC Ground Ring* (Perfect Conductor Ground Ring): (Default) The tool creates a ground ring around the selected objects, as shown below, and connects the ports to the ring.



Related Topics

[Specifying Die Ground Settings for an IC Layout]

[LVS Resistors in Electromagnetic Simulations]

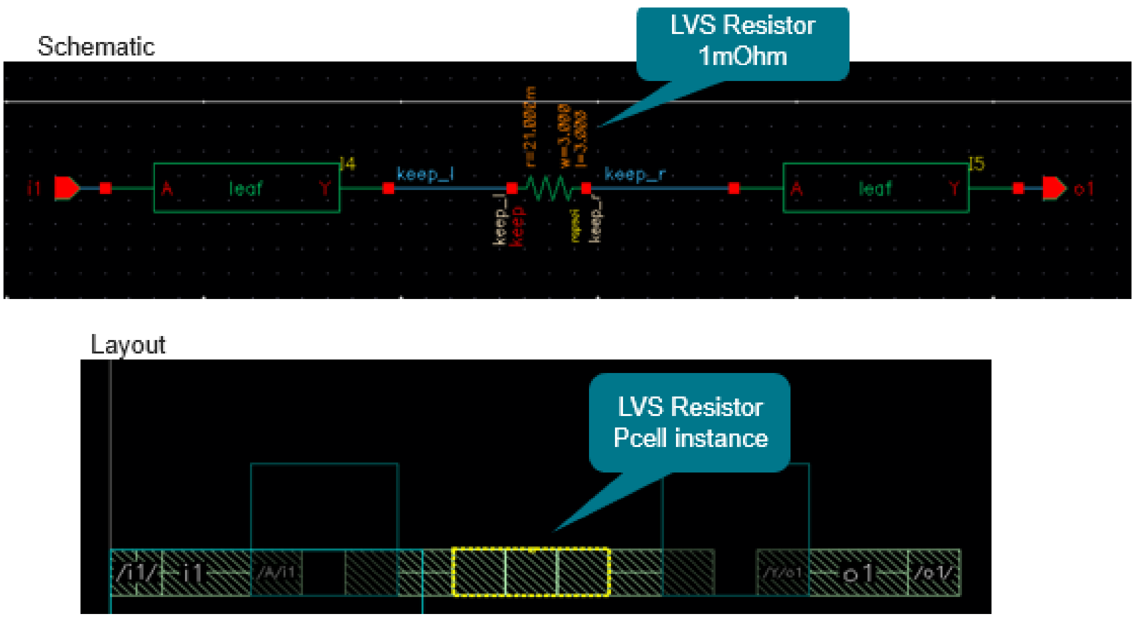
© 2023 Cadence Design Systems, Inc.

LVS Resistors in Electromagnetic Simulations

If a selected net contains an LVS resistor, a resistor with a vary small resistance value, depending on how the resistor is placed in the schematic or its properties are set in Virtuoso, the information about the LVS resistor is also sent to the electromagnetic simulator along with the details of the net to which it is connected.

Virtuoso considers LVS resistors that are added in the following three scenarios:

Scenario 1: The LVS resistor is added as an instance in both the schematic and the layout

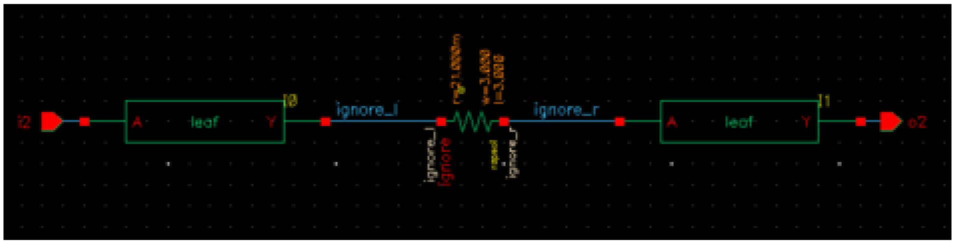


In this scenario, you can select the LVS resistor like any other instance. If you select it on the layout canvas and add to the model, the LVS resistor is listed on the *Selection* tab and sent to the electromagnetic solver along with other nets and instances. It is also saved as a part of the S-param file saved after simulation.

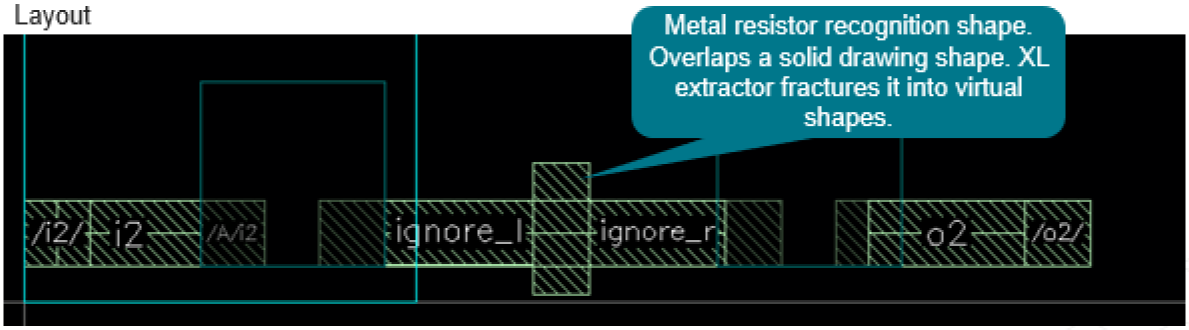
© 2023 Cadence Design Systems, Inc.

Scenario 2: The LVS resistor is a cut shape on an LPP that is defined as a stopping layer for the connectivity extractor

Schematic



Layout

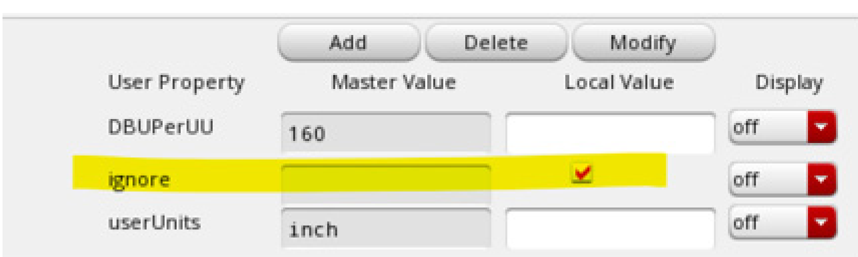


The net to which the resistor is attached is split in both the schematic and the layout views. In the example figure shown above, the resistor named `ignore` is placed in the schematic and the net is split into two parts, `ignore_l` and `ignore_r`. Only the schematic view contains the resistor as an instance. The layout view shows it as a virtual metal resistor recognition shape. This layer is defined as a stopping layer in the technology file. For details, see Specifying Stop Layers in the `validLayers` Constraint.

In this case, you should define the resistor as ignored in the Layout XL bindings. It will then be automatically removed when the extracted view is generated.

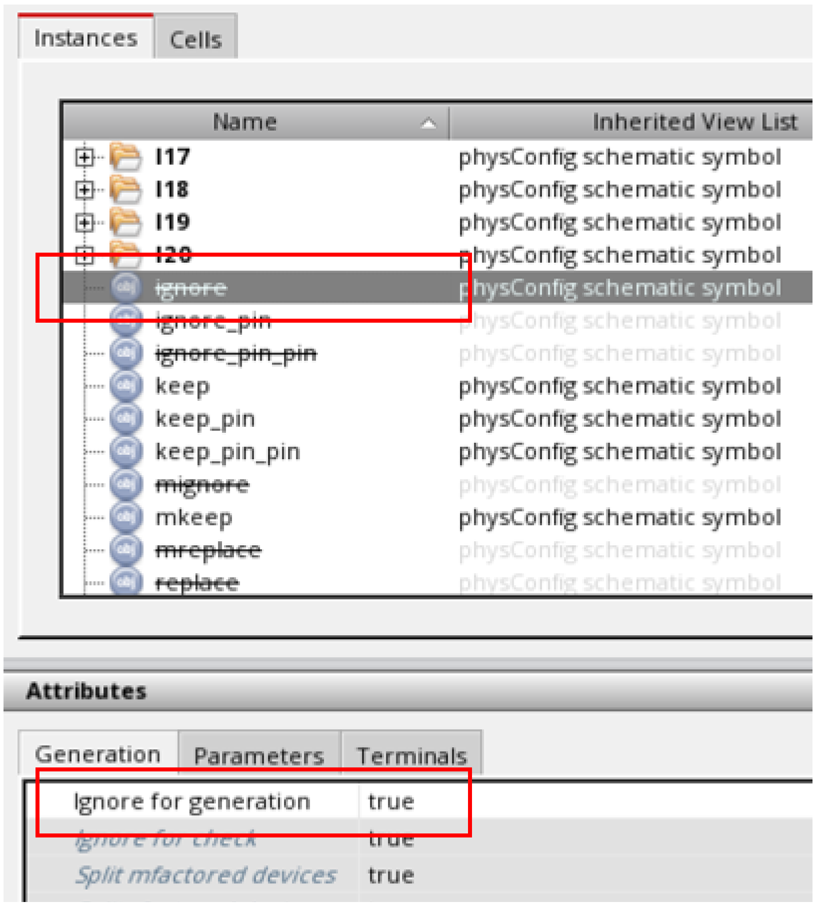
To ignore the resistor instance, do one of the following:

- In the Edit Object Properties form for the schematic instance, set the *ignore* property.



© 2023 Cadence Design Systems, Inc.

- To ignore a specific instance of an LVS resistor, on the *Instances* tab of the Configure Physical Hierarchy window, set the *Ignore for generation* property for that resistor instance to `true`.



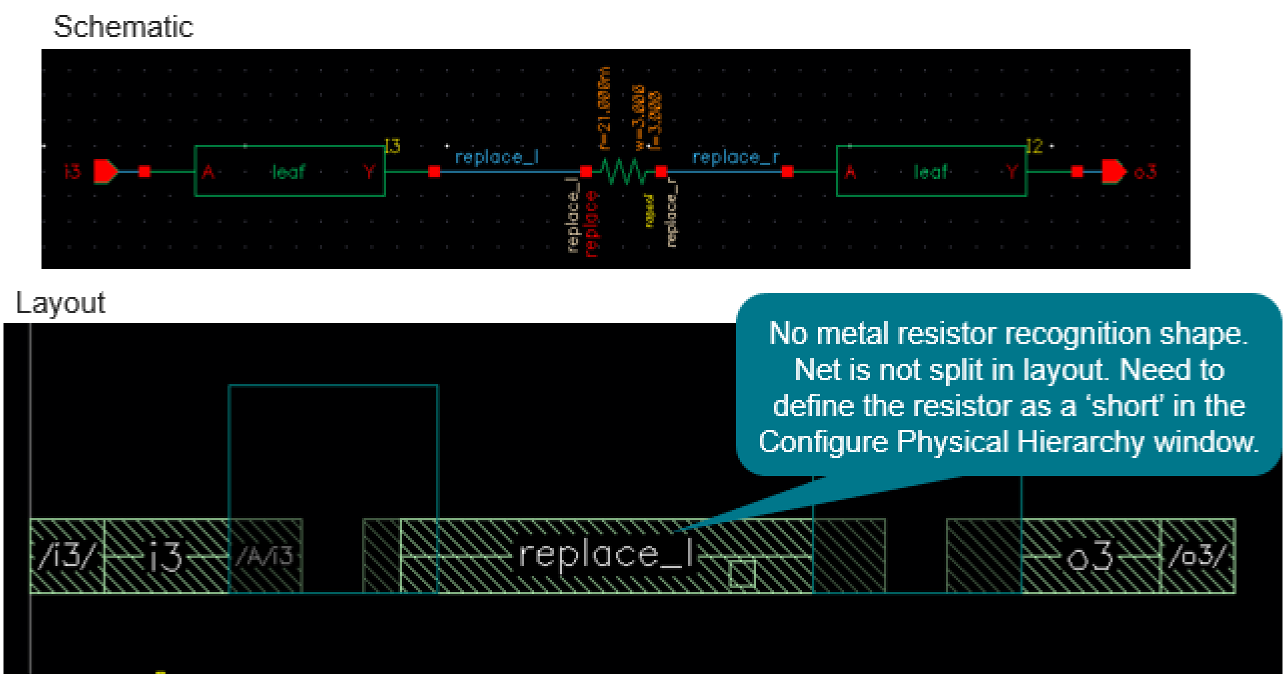
- To ignore all instances of an LVS resistor, set the *Ignore for generation* property for the resistor cell to `true` on the *Cells* tabs of the Configure Physical Hierarchy window.

  For more details, refer to Ignoring a Cell for Generation or Check in *Virtuoso Layout Suite XL: Connectivity Driven Editing User Guide*.

When you configure these settings, the tool ignores the LVS resistor for electromagnetic simulation, but lists it as an embedded component in the header of the S-param file.

© 2023 Cadence Design Systems, Inc.

Scenario 3: The LVS resistor is a shape that is ignored by the connectivity extractor

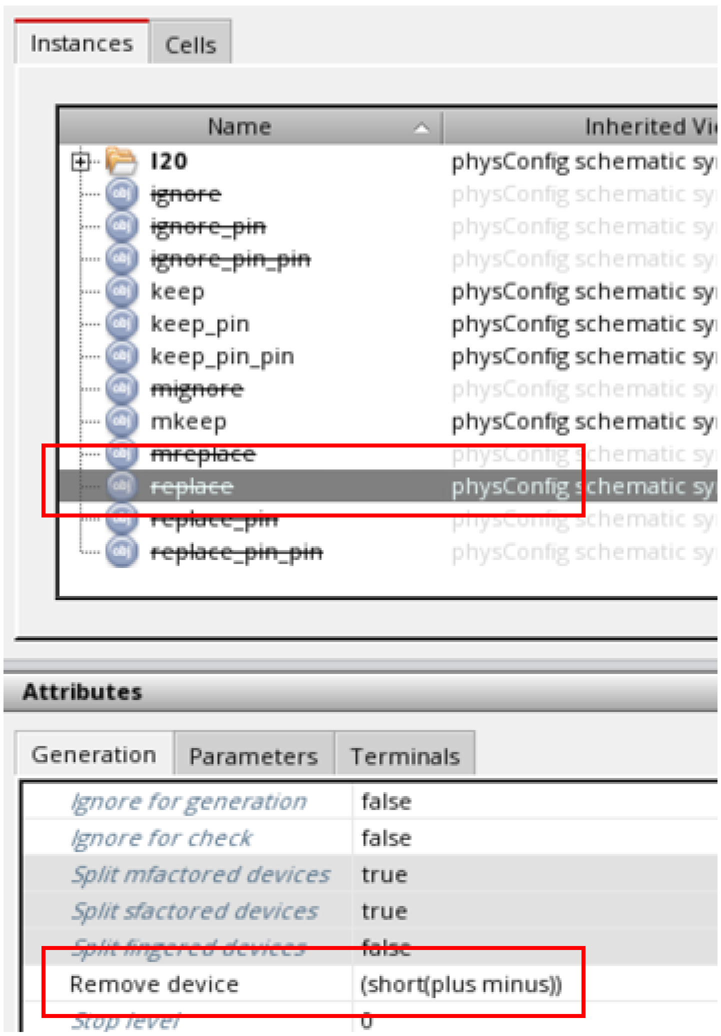


The net to which the LVS resistor is connected is split only in the schematic view. In the example shown above, the resistor named `replace` is placed in the schematic and the net is split into `replace_l` and `replace_r`. Both parts of the net in the schematic are bound to the common net in the layout, which is `replace_l` in this example.

In the layout, there is either no metal resistor recognition shape at all or it is on an LPP that is not defined as a stopping LPP by the XL extractor and is, therefore, ignored by the connectivity extractor.

© 2023 Cadence Design Systems, Inc.

In this scenario, you can ignore the resistor by specifying it as a short on the *Instances* tab of the Configure Physical Hierarchy window.



In this example, `plus` and `minus` are the names of the pins of the LVS resistor.

The short devices are ignored during electromagnetic simulation and are specified as embedded components in the header of the S-param file.

For more details, refer to the Remove Device option in the *Virtuoso Layout Suite XL: Connectivity Driven Editing User Guide*.

© 2023 Cadence Design Systems, Inc.

Removing Selected Objects

To remove any instance or net from the selection:

1. Click one or more objects in the *Object Selection* section on the *Selection* tab.
2. Click *Delete instances/nets* ( ) on this tab.

   The objects are removed.

© 2023 Cadence Design Systems, Inc.

Specifying Die Ground Settings for an IC Layout

The die ground setting defines the size of the metal shape that is created at the bottom of the substrate. This also establishes the X and Y extents of the 3D box that will be used for the simulation region by Clarity 3D Solver.

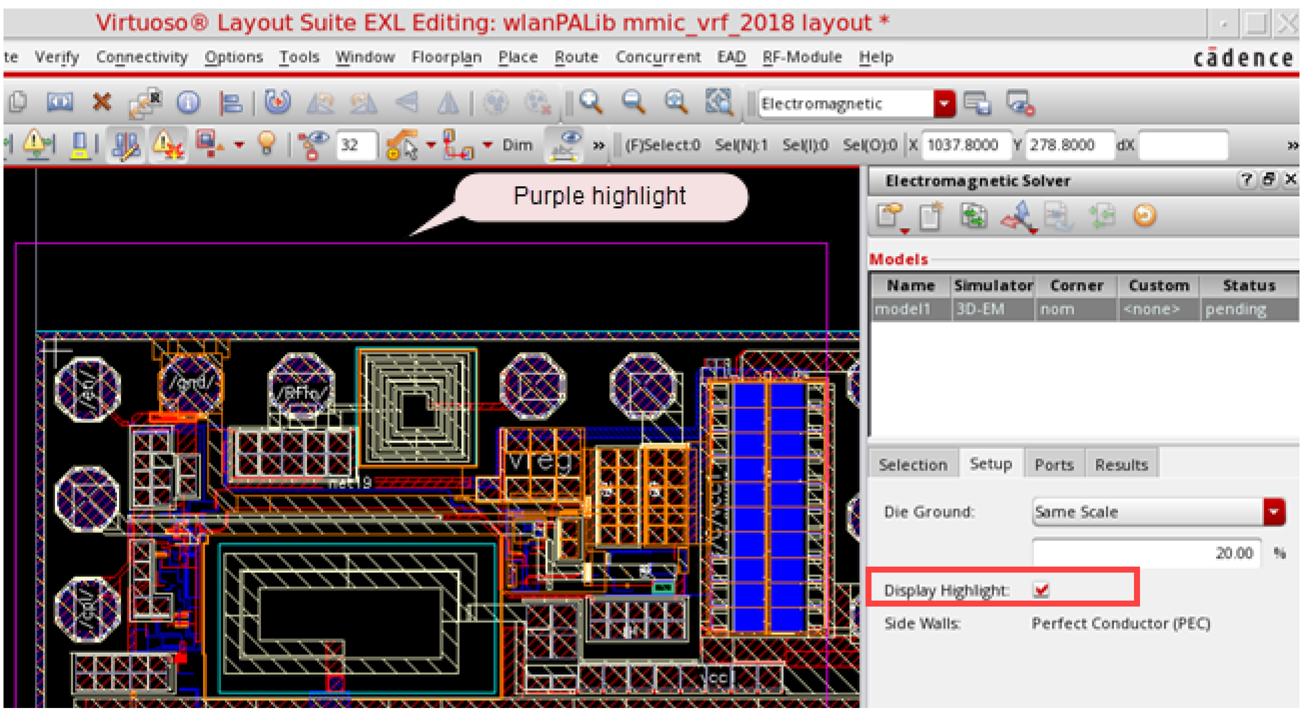
To define a die ground:

1. On the *Setup* tab, specify the scale type and scale percentage in the *Die Ground* field.

   You can specify the scale type either as a percentage of the width or height of the model or as an absolute dimension in micron. This scale factor is applied around the bounding box of the model, which is the bounding box of all selected nets and instances, optionally trimmed by a cutting boundary.

2. Select the *Display Highlight* check box.

   This displays a flashing purple square on the layout canvas to represent the X and Y extents of the box to be used as the simulation region, as shown in the figure of an IC layout shown below.



The *Side Walls* setting on the *Setup* tab is dependent on the *Current Return Path* setting on the *Selection* tab.

© 2023 Cadence Design Systems, Inc.

Related Topics

Selecting Objects from an IC Layout

© 2023 Cadence Design Systems, Inc.

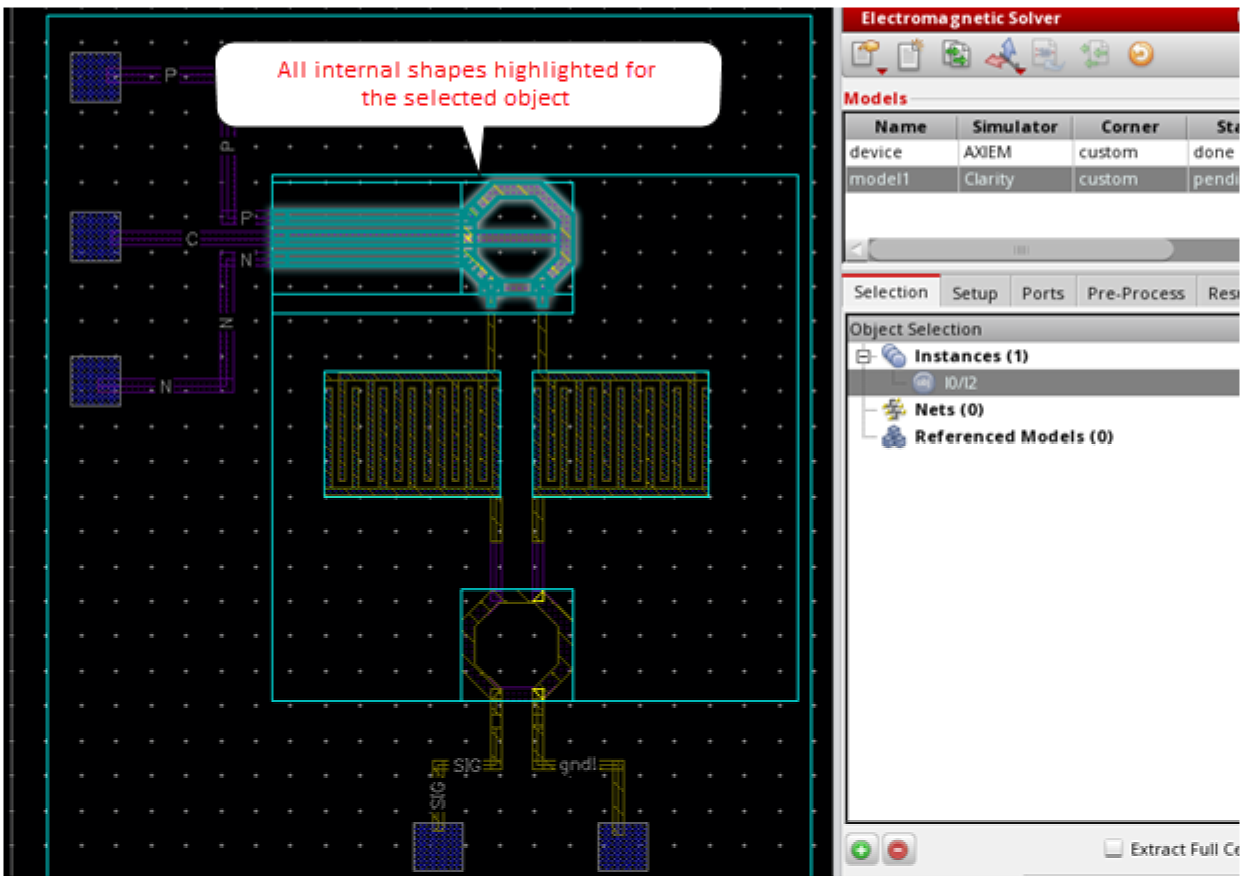
Highlighting Selected Objects on the Layout Canvas

To locate an object selected for a model:

➡ Click the object in the *Object Selection* section.

Alternatively, you can also use the *Select All* or *Deselect All* commands from the context-sensitive menu of the objects on this tab to highlight all objects or remove all highlights.

Layout MXL highlights the object and all the shapes inside it. This lets you identify the objects and shapes that would be sent to the solver.



The highlighting of shapes in the hierarchy is dependent on the display stop level set for the layout.

Related Topics

[Selecting Objects from an IC Layout]

© 2023 Cadence Design Systems, Inc.

Validating a Flip-Chip Die in a Package Layout

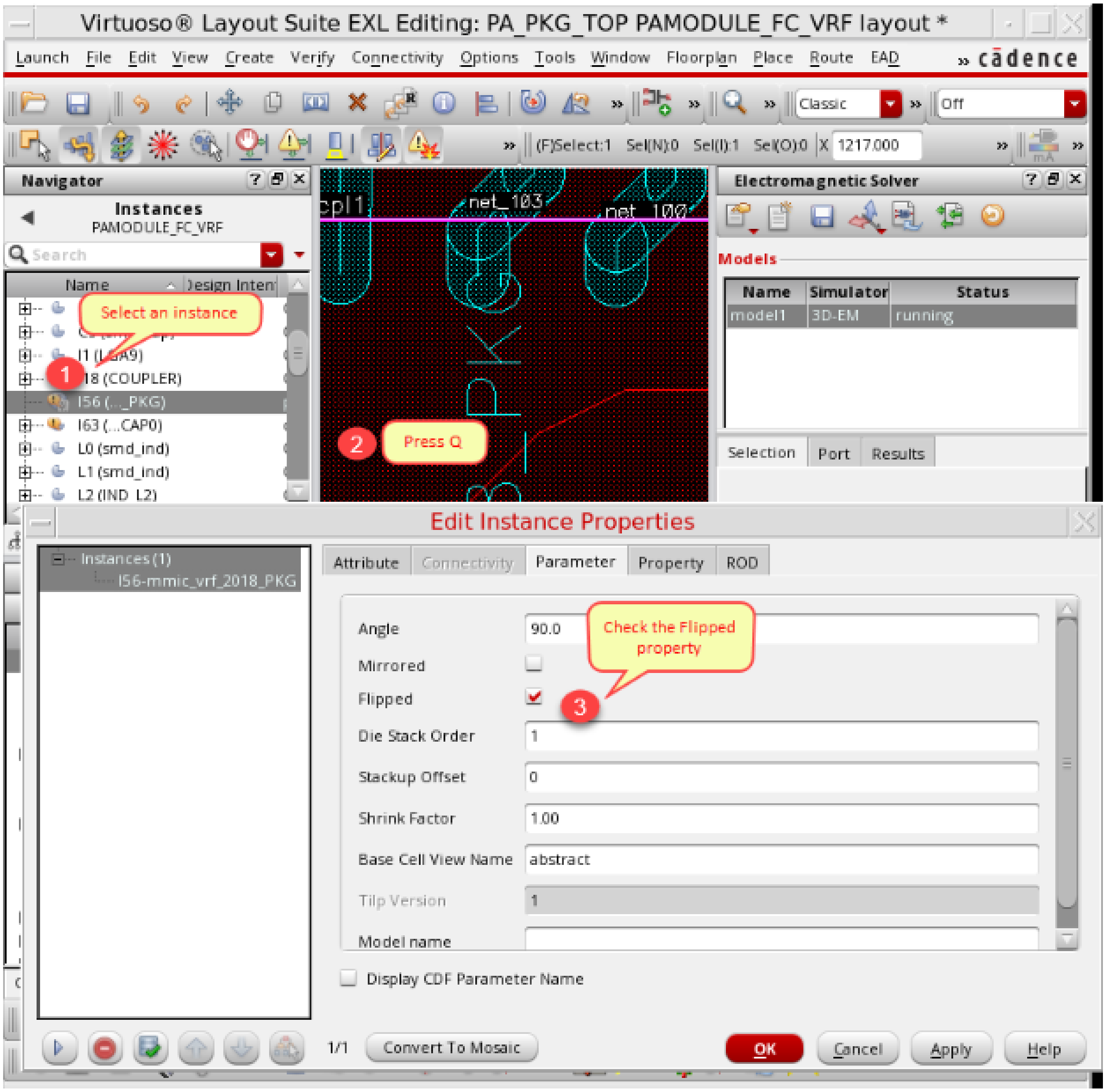
If you have a flip-chip die in the package design, ensure that it is identified correctly. For this, review the properties of the instance on the Edit Instance Parameters form and ensure that the *Flipped* parameter is selected.

To validate a flip-chip die:

1. Select the instance of a flip-chip die in the layout.
2. Press the `Q` key to open the Edit Instance Properties form.

© 2023 Cadence Design Systems, Inc.

3. Ensure that the *Flipped* check box is selected in the Edit Instance Properties form.



Related Topics

[Selecting Objects from an IC Layout]()

© 2023 Cadence Design Systems, Inc.

Port Creation for Clarity Models in IC Layouts

After object or reference model selection, an important task is to specify ports for the selected objects. You can specify ports on the *Ports* tab of the Electromagnetic Solver assistant.

For an IC layout, the Electromagnetic Solver assistant lets you create ports in two ways:

- By using the inbuilt automatic port generation feature
- By manually adding ports and customizing their settings

Related Topics

Creating Ports Automatically for Clarity Models in IC Layouts

Creating Ports Manually for Clarity Models in IC Layouts

Port Management for Clarity Models

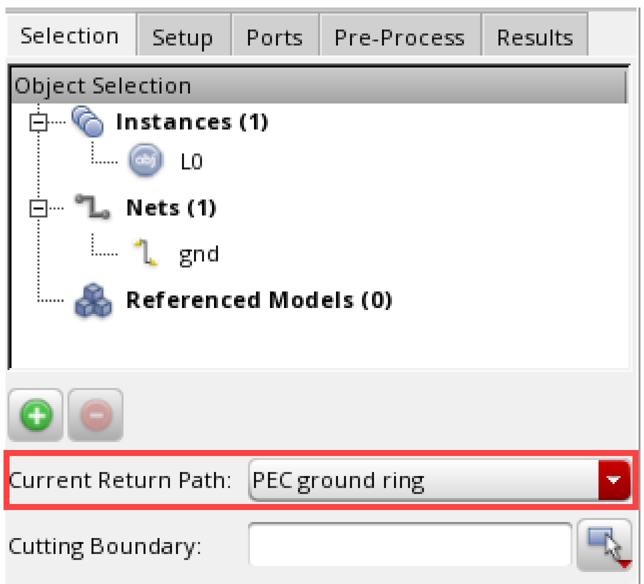
© 2023 Cadence Design Systems, Inc.

Creating Ports Automatically for Clarity Models in IC Layouts

The automatic port generation feature uses some pre-defined rules to create ports for the selected objects and nets. You can use this feature when you are not sure about the negative edges of ports, and you want Layout MXL to automatically identify the current return path.

To create ports for an IC layout by using the automatic port generation feature:

1. Ensure that all instances and nets in the model can be enclosed in a perfectly conducting box on all sides.
2. On the *Selection* tab, choose `PEC Ground Ring` from the *Current Return Path* drop-down list.



3. On the *Setup* tab of this assistant, select an appropriate option from the *Die Ground* drop-down list and specify a size or scaling option for the die ground.

   A die ground is a metal shape that will be inserted into the model on an automatically generated layer located below the substrate. The conductivity of the die ground is very high. This non-physical conductivity might result in warning messages from the solver, but you can safely ignore those warnings.

4. On the same tab, select the *Display Highlight* check box. This helps in viewing an accurate representation of the port placement.

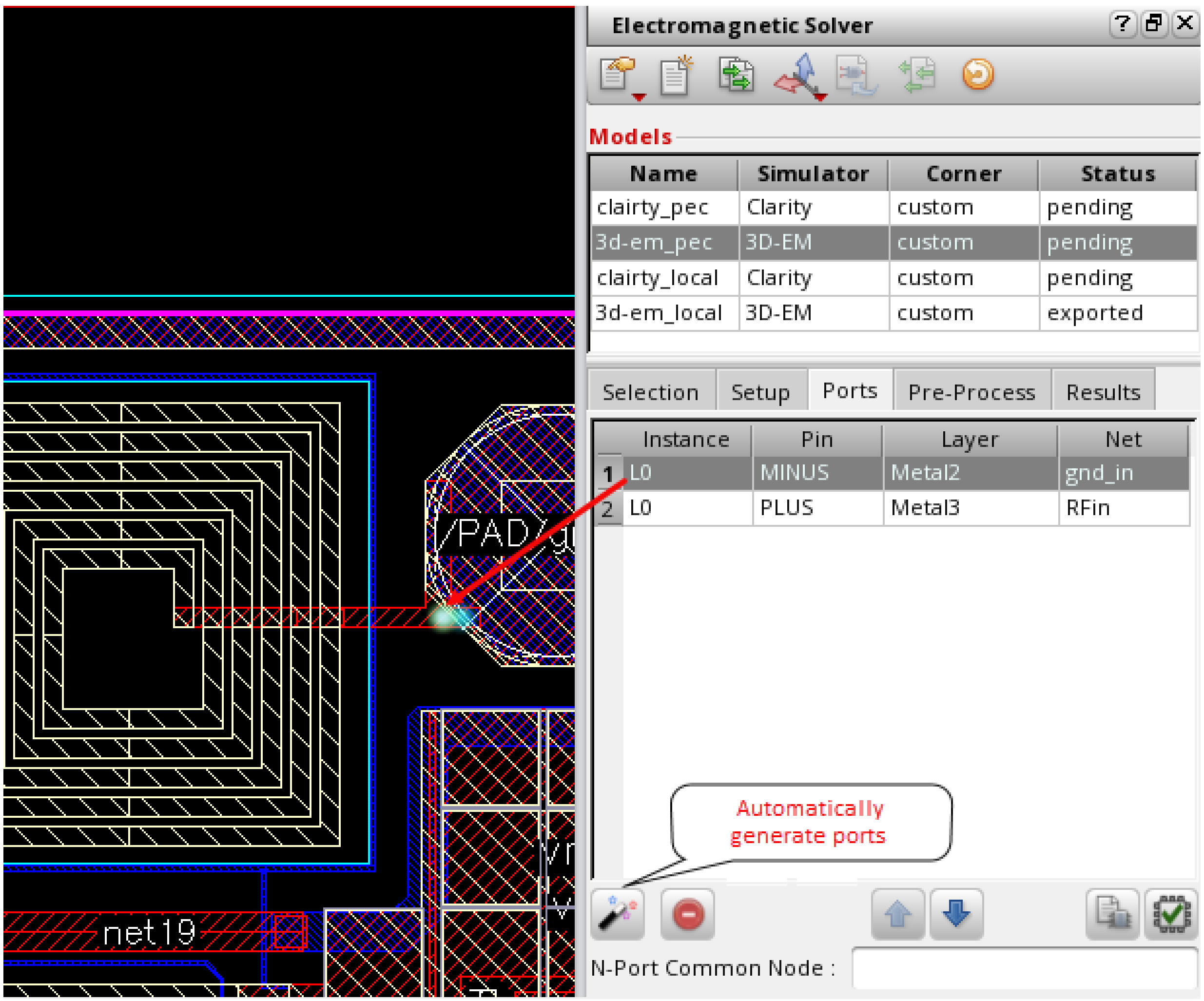
   The *Side Walls* field automatically shows a value depending on the current return path set for the model.

© 2023 Cadence Design Systems, Inc.

5. On the *Ports* tab, click *Automatically Generate Ports* to automatically generate ports for the selected objects.

   Layout MXL identifies ports of the selected instances. For each port, it adds an extension towards the boundary and a port at the end of each extension. The + edge is on the extension and the - edge is on the PEC boundary.

   The details of ports are displayed in the table on the *Ports* tab.



   The blue dots indicate the negative connection of the ports on the PEC boundary. The green dot indicates the other end, which is the positive connection of the port extension. All the ports for an instance are on the same layer as that of the instance.

6. If you do not need any port, select its row, and click *Remove Port*.

7. Specify a net name in the *N-Port Common Node* field. The reference pin for the N-port instance in the schematic is connected to this net.

© 2023 Cadence Design Systems, Inc.

Note: If an instance selected for a model is listed in the value of the setNotEmbedded environment variable it has pins, the tool generates ports on its pins.

Related Topics

Ports Tab (EMX)

Guidelines for Current Return Path in Models that Use Clarity 3D Solver

Creating Ports Manually for Clarity Models in IC Layouts

© 2023 Cadence Design Systems, Inc.

Creating Ports Manually for Clarity Models in IC Layouts

If you know where to place the negative edges of the ports and you want full control on the ports, select objects for a model and proceed with the following steps:

1. On the *Selection* tab, set the *Current Return Path* field to `Local Reference`.
2. On the *Ports* tab, specify the name of a reference net in the *Reference Nets* field.
3. On the *Setup* tab of this assistant, select an appropriate option from the *Die Ground* drop-down list and specify a size or scaling option for the die ground.

   A die ground is a metal shape that will be inserted into the model on an automatically generated layer located below the substrate. The conductivity of the die ground is very high. This non-physical conductivity might result in warning messages from the solver, but you can safely ignore those warnings.
4. On the same tab, select the *Display Highlight* check box. This helps in viewing an accurate representation of the port placement.

   The *Side Walls* field automatically shows a value depending on the current return path set for the model.
5. (Optional) On the *Ports* tab, click *Automatically Generate Ports* to automatically generate ports for the selected objects.

   Layout MXL identifies ports of the selected instances and populates the ports table with their details. You can review and modify the ports as per the requirements of your design.

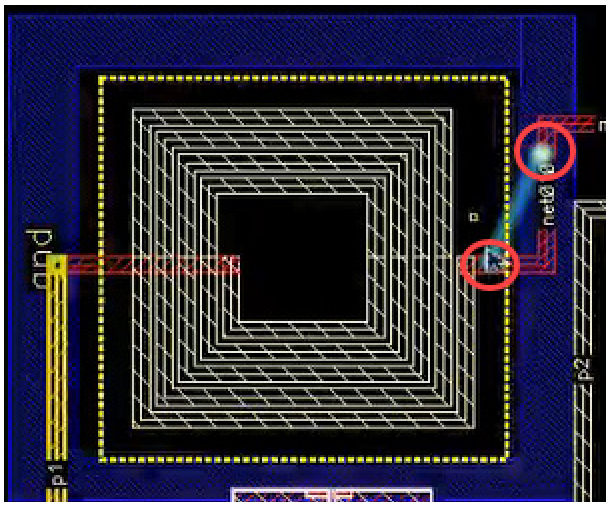
   If you do not want the tool to create any ports so that you get full control on the ports created for the model, skip this step.
6. Click *Add Port* to create a new port.

   A new row is added to the ports table. For each port, two rows are added in the edges table below the ports table. These rows show the details of connections of the port.
7. In the row for the newly created port, select an appropriate port type in the *Type* column.

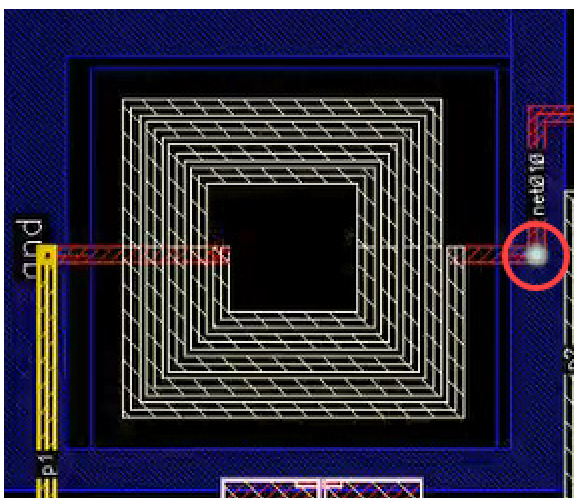
   The possible port types are:

© 2023 Cadence Design Systems, Inc.

- `Same Layer`: The positive and negative edges for these ports are placed horizontally on the same layer.

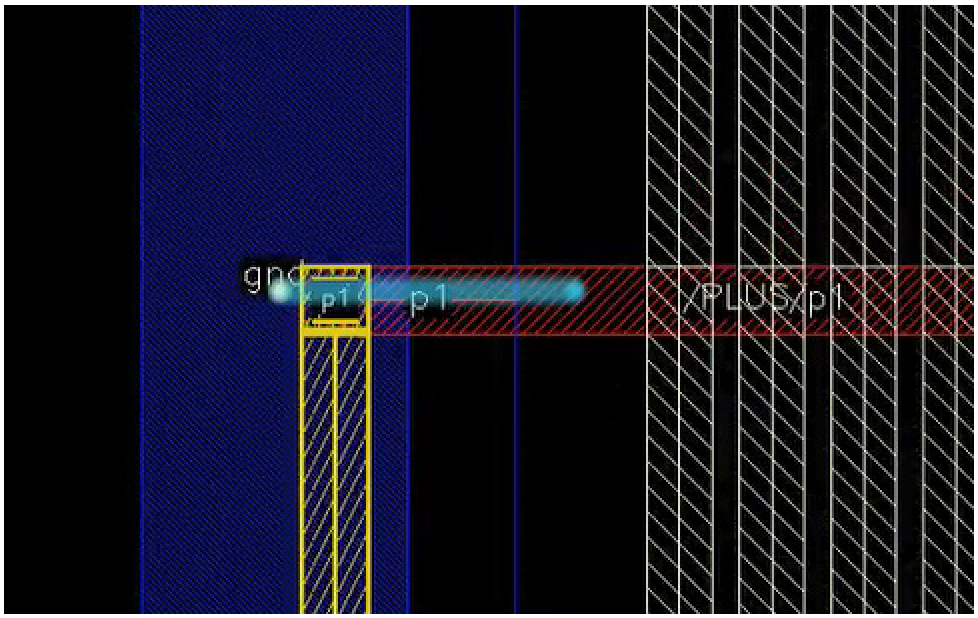


- `Cross Layer`: The positive and negative edges for these ports are placed on different layers. These ports connect the layers vertically.



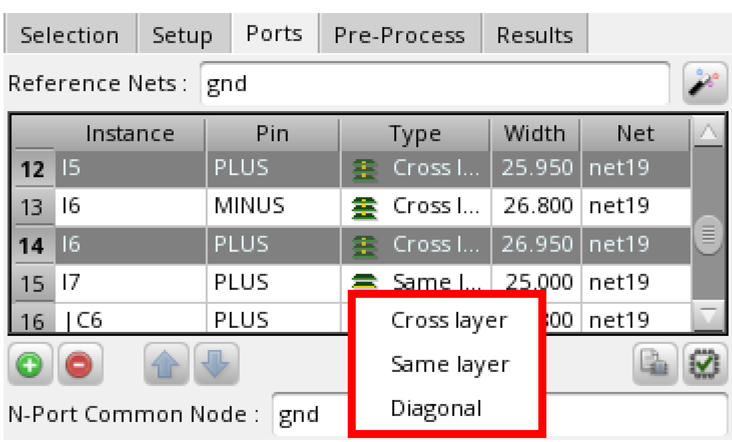
© 2023 Cadence Design Systems, Inc.

- `Diagonal`: The positive and negative edges for these ports are placed diagonally on different layers. This type of port can be created for Clarity solvers only.



The diagonal connections can be seen in the Clarity environment as connecting two layers diagonally.

To specify the port type for multiple ports, select the rows for the ports, right-click and choose *Set Port Type*. A list of the possible port types is displayed at the pointer location. Choose the port type you want to set. All the rows are updated with the selected port type.



8. If required, modify the port width in the *Width* column.

© 2023 Cadence Design Systems, Inc.

To set the same width value for multiple ports, select the rows for those ports, right-click and choose *Set Port Width*. In the form that is displayed, specify a width value, and click *OK*.

9. Click each port and review the edges in the edges table. If required, you can edit the edges.

Related Topics

Port Management for Clarity Models

© 2023 Cadence Design Systems, Inc.

Port Management for Clarity Models

Meaningful electromagnetic simulation results depend greatly on the quality of port definitions. If the port definitions are not correct for your layout, you may get results, but many times, they cannot be trusted. Therefore, it is important to validate them and make modifications, where required.

The *Ports* tab of the Electromagnetic Solver assistant provides various commands to check the ports and to change the placement or alignment of the ports.

Related Topics

Managing Edges for Ports Created for Clarity Models

Validating Ports Created for Clarity Models

Grouping Ports Created for Clarity Models

© 2023 Cadence Design Systems, Inc.

Managing Edges for Ports Created for Clarity Models

To manage the edges for a port:

1. Click the row for that port in the ports table.

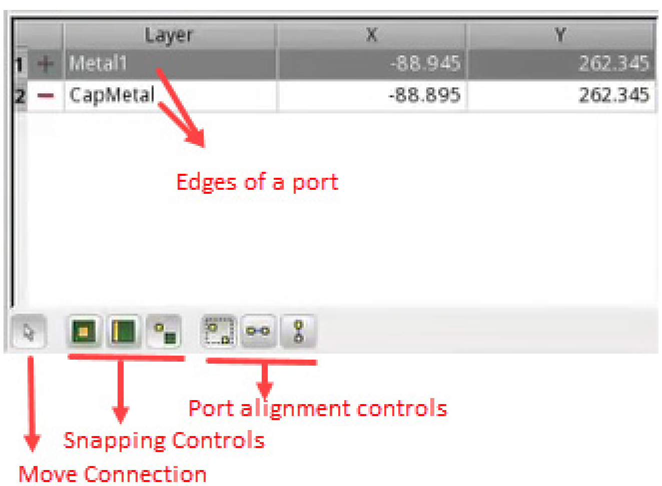
   Check that the X and Y locations for the positive and negative ends are different, and if the location is same, they are on different layers.

2. (Optional) Change the layer name in the *Layer* column to place the ports on other layers.

3. If required, move the coordinates for a port connection by selecting its row and clicking *Move Connection* at the bottom.

   By default, the X and Y coordinates of these connections are set to `0`.

   The *Move Connection* command displays the controls for port snapping and alignment, as shown below.



   The three snapping controls are snap to the center of the shape, snap to the edge of the shape, and no snapping, respectively.

   The three controls for alignment adjustment are no alignment, align horizontally (move at the same Y coordinate), and align vertically (move at the same X coordinate), respectively.

4. Choose a snapping control and an alignment control and move the pointer on the canvas to move the positive or negative edge of the port.

© 2023 Cadence Design Systems, Inc.

Validating Ports Created for Clarity Models

To validate all the ports in the ports table:

➡ Click *Validate Ports* on the Ports tab.

Virtuoso validates all ports. If the ports do not follow the rules, it displays warning messages in CIW.

A port is considered as valid when the following criteria are met:

- The end edges of a port are fully enclosed in metal
- The port is not touching any other port or metal between the end edges
- The port does not touch or overlap any other port

Running port validation in the Electromagnetic Solver assistant ensures that these errors are not reported by the simulator. If you do not validate the ports in Virtuoso, the EM simulator identifies these issues, and fails to run simulations.

© 2023 Cadence Design Systems, Inc.

Grouping Ports Created for Clarity Models

You can group ports with the same electrical potential so that the simulator considers them as a single port. Grouping improves simulation performance by reducing the actual number of ports to be managed by the simulator.

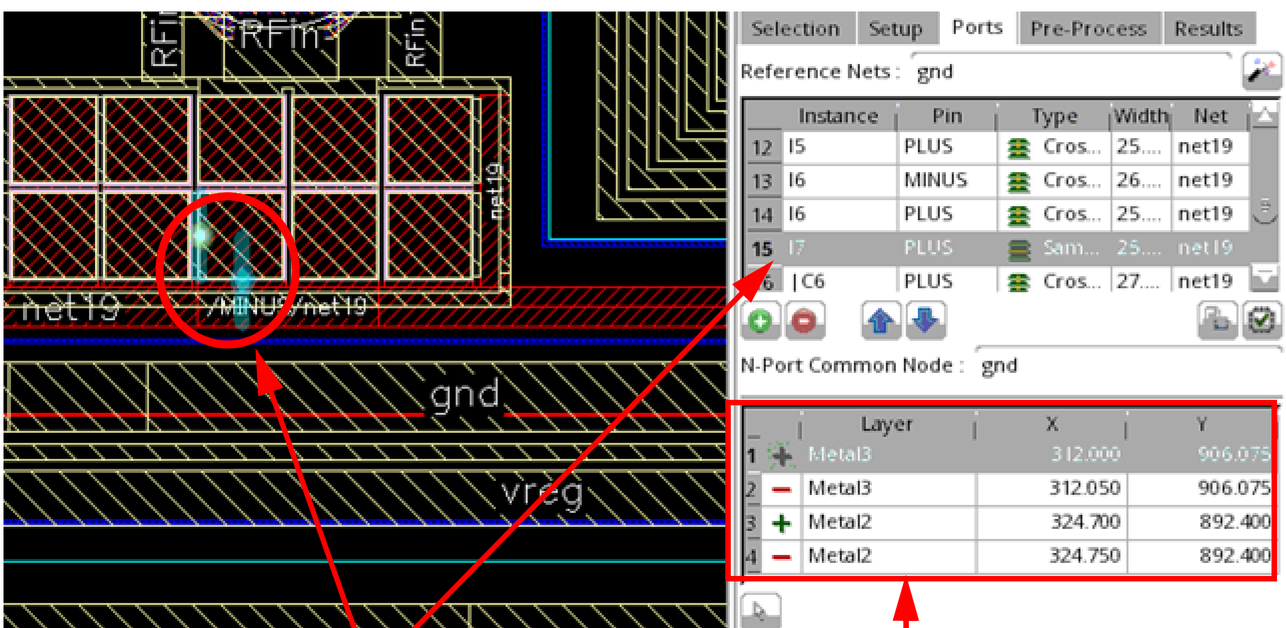
To group ports:

1. Select two or more ports in the ports table.
2. Right-click and choose *Group Ports*.

   The grouped ports are collapsed into one.

Observe the following for the port group:

- Only the top most port in the selected set is visible and other ports are hidden.
- The edge details of all the ports in the port group are displayed in the edge details of the visible port.
- When you select the port group in the *Ports* tab, all the ports in the group are highlighted in the layout canvas.



© 2023 Cadence Design Systems, Inc.

To dissolve a port group, right-click the group in the ports table and choose *Ungroup Port*. The rows for ports that were grouped are displayed at the end of the ports table.

© 2023 Cadence Design Systems, Inc.

Guidelines for Current Return Path in Models that Use Clarity 3D Solver

You can specify the current return path for a model in the *Current Return Path* field on the *Selection* tab of the Electromagnetic Solver assistant. The current return path is an important consideration when deciding about the approach to create ports for a model.

If you know where to place the negative edges of the ports and you want full control on the ports, choose *Local Reference* on the *Selection* tab. You can begin with automatic port generation, but you can modify the settings or create new ports as per your requirements.

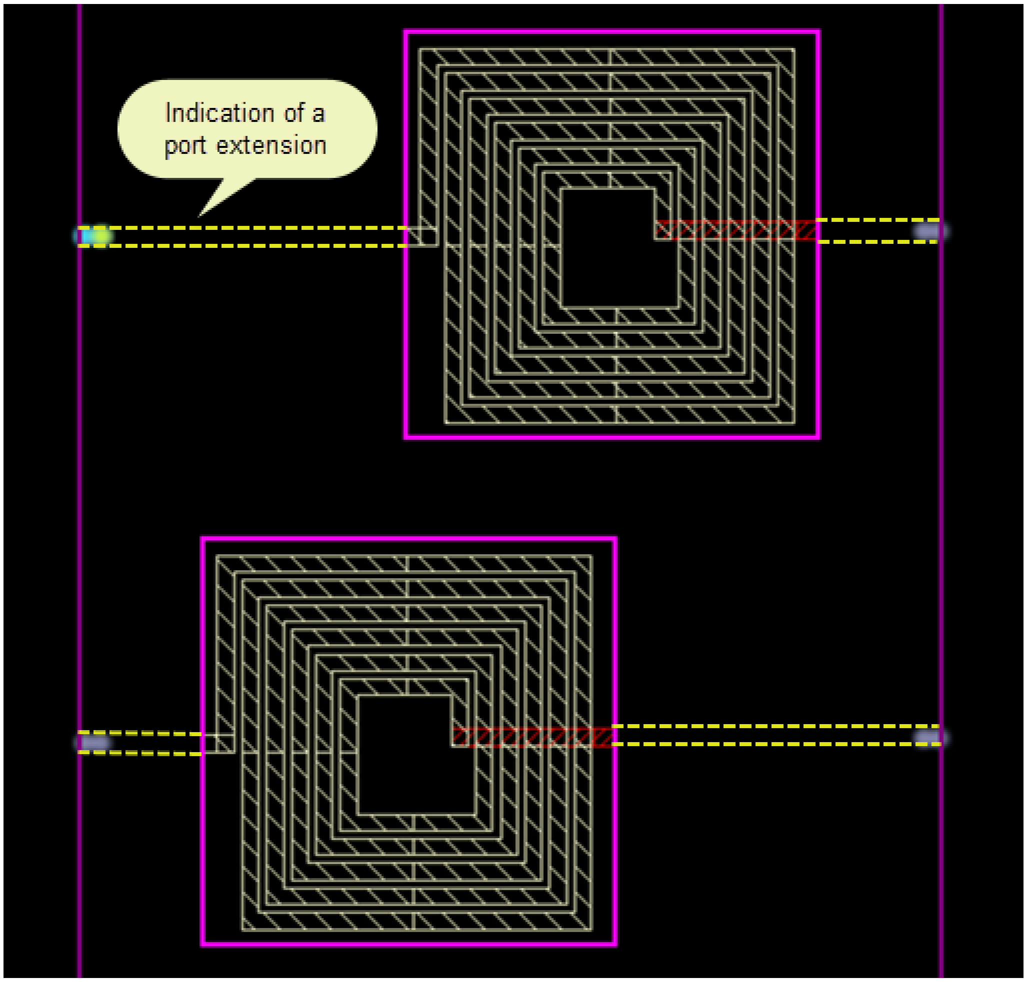
If you are not sure about the negative edges of ports, and you want Layout MXL to automatically identify the current return, ensure that the model can be enclosed in a perfectly conducting box on all sides, and choose *PEC Ground Ring* and proceed with automatic port generation.

You can consider the following guidelines while choosing a current return path:

- The `Local Reference` current return path can be used in all possible cases and there is no limitation for the length and alignment of port extensions, but you need to find the correct ground for each port by connecting it to the real ground metal or plane below it or next to it.
- The *PEC Ground Ring* option works best for simple passives, such as inductors, when the instances can be placed within a boundary. Placing port extensions might not be feasible in complex scenarios because extensions that cross over each other or other metals can result in shorts.
- Clarity 3D Solver supports only rectangular deembedding zones. When choosing *PEC Ground Ring*, ensure that all the ports are aligned to the boundary with the same length and the same starting point. Otherwise, deembedding is not perfect. If the above-mentioned requirements are not met, it is recommended to choose *Local Reference* as the current return path.

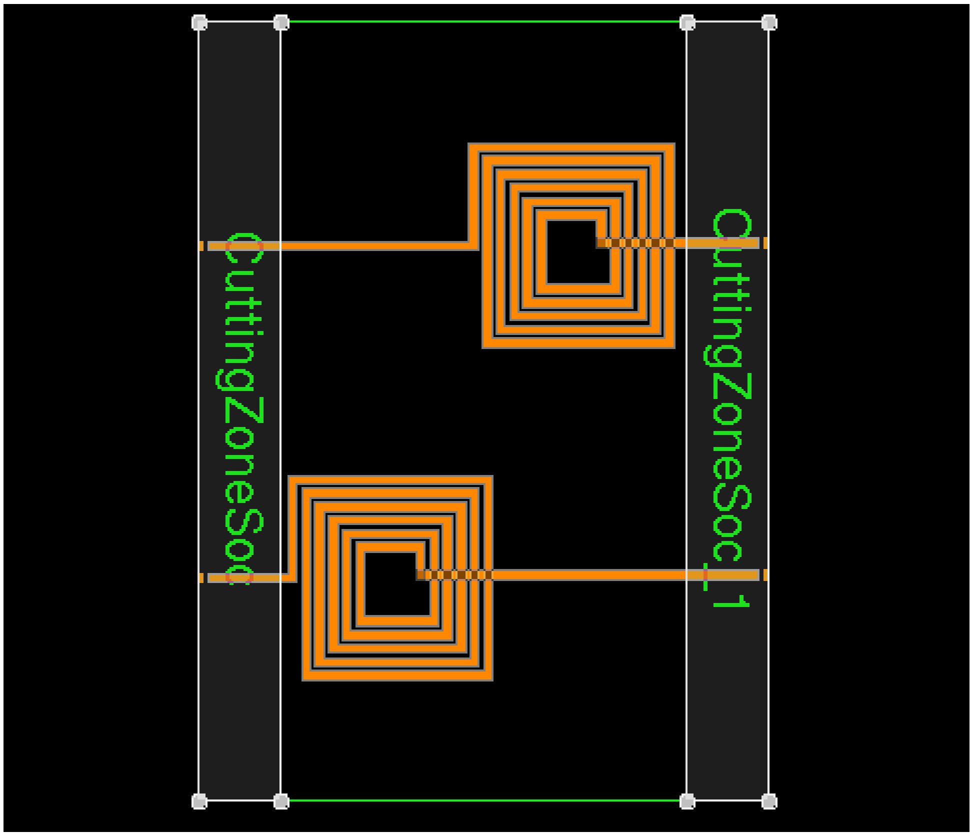
© 2023 Cadence Design Systems, Inc.

Consider the following example where the left port for the upper instance needs to be placed on the left boundary edge even if it is far away from the instance.



© 2023 Cadence Design Systems, Inc.

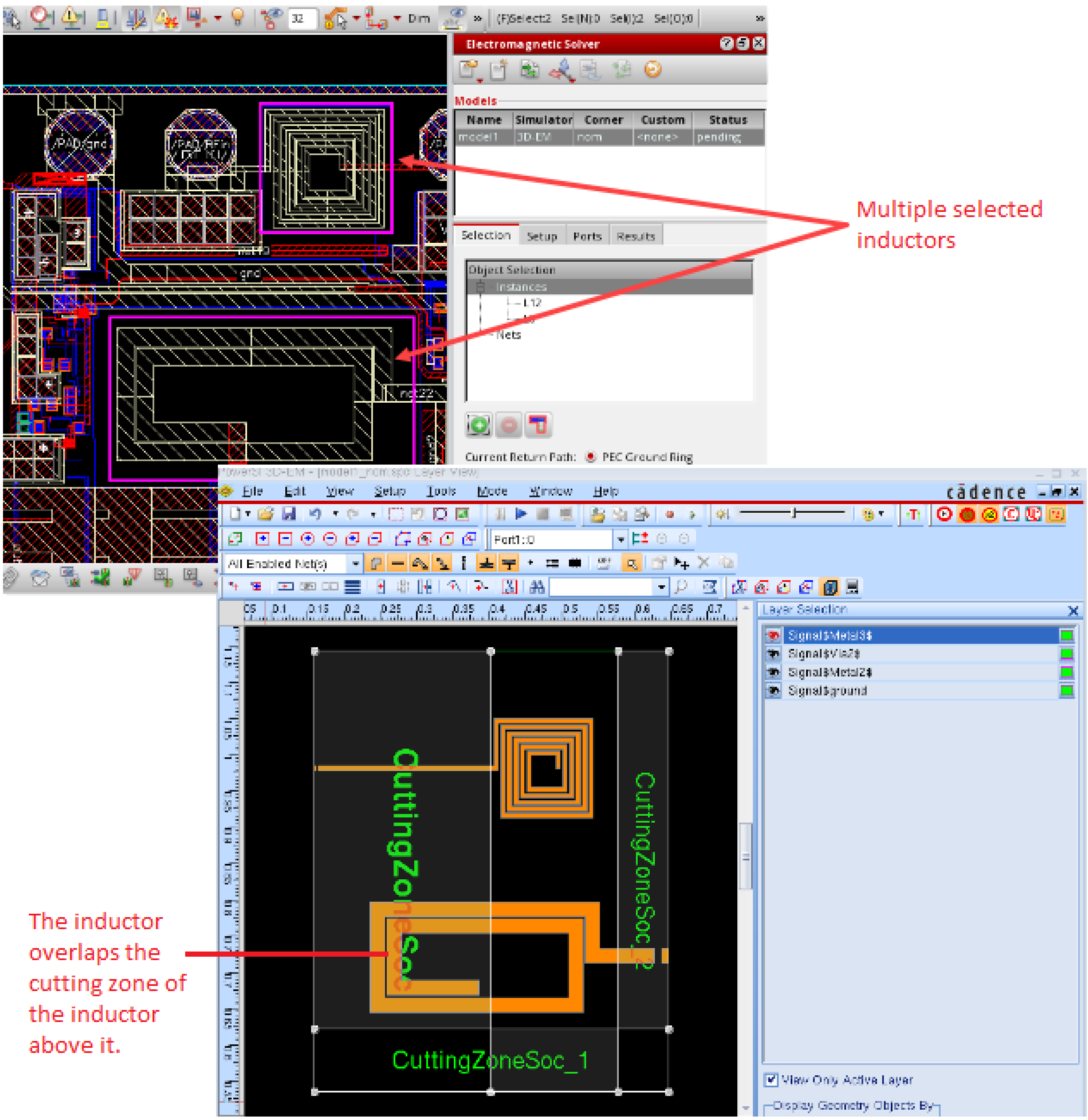
When deembedding is done for the above example, the cutting zones are properly placed in such a way that only port extensions are eliminated from the model, as shown below.



If the port extensions are of different sizes, the deembedding zone needs to be wide enough to cover the longest port extension of one device. In that case, there are chances

© 2023 Cadence Design Systems, Inc.

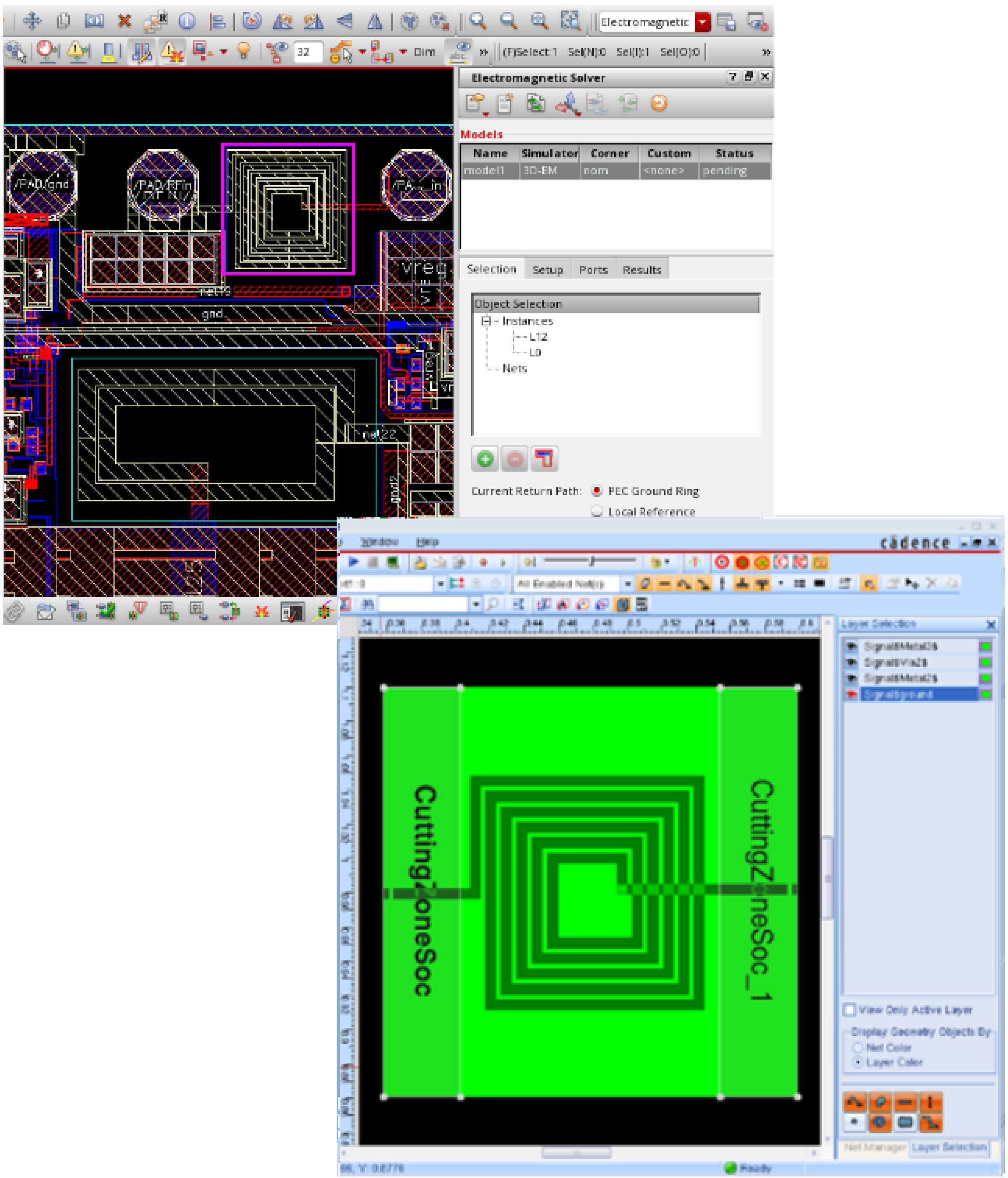
that the cutting zones cover a part of real geometry of other devices, as shown in the example given below.



This is not a desired case. The deembedding zone is a zone where the effect of port extensions is calculated and removed from the model. If a real geometry is falling in the deembedding zone or cutting zone, as shown above, it results into the loss of the device details from the model. Only the effect of a part of device that is falling in the non-cutting zone is considered for model creation.

© 2023 Cadence Design Systems, Inc.

Therefore, it is necessary to create the model in such a way that the cutting zones correctly overlap the port extensions on all sides, as shown in the figure given below.



© 2023 Cadence Design Systems, Inc.

Specifying Ports for Clarity Models in Package Layout

The settings required for ports in a package layout are different from those in an IC layout. Therefore, the *Ports* tab for a package layout shows different fields.

For a package layout, you do not require die ground settings, so the *Setup* tab that appears to the left of *Ports* tab for IC layouts is not available for package layouts. The Ports tab contains two sub-tabs, *Setup* and *Components*.

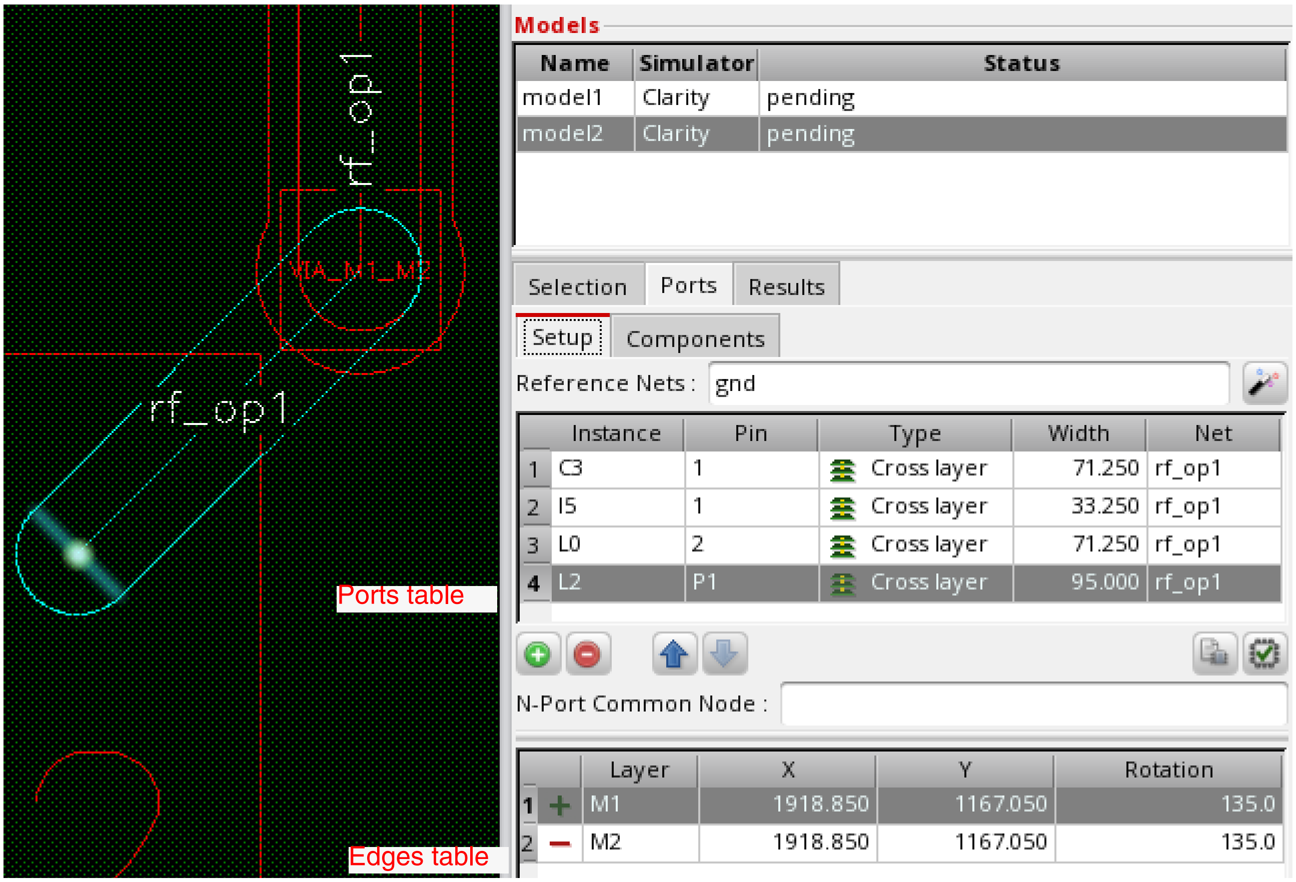
To create ports for a package design:

1. Select instances and nets to be included in the model.
2. On the *Setup* sub tab of the *Ports* tab, specify a supply or ground net in the *Reference Net* drop-down list.
3. To automatically generate ports for the selected instances, click  *(Automatically Generate Ports*) to the right of *Reference Nets*.

   Layout MXL populates both the sub-tabs in the *Ports* tab as described below.

© 2023 Cadence Design Systems, Inc.

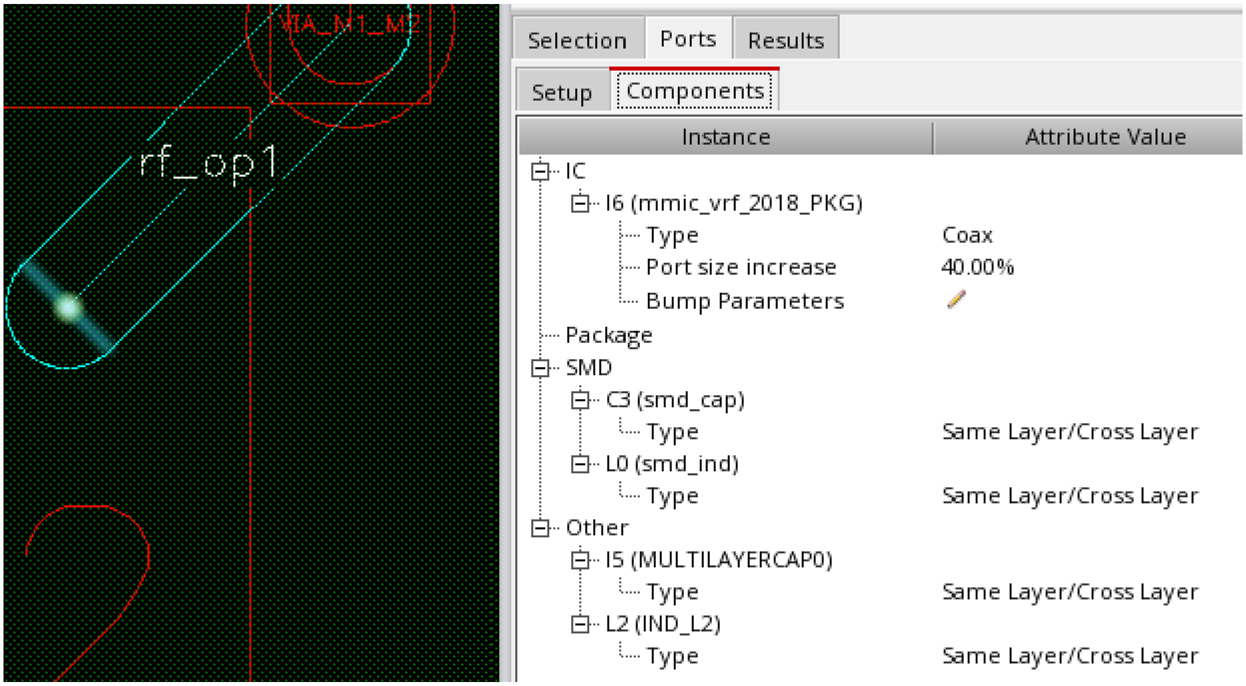
In the *Setup* sub-tab, the tool identifies ports for the selected instances and loads their details in the table. For each port, the positive and negative edges are also shown.



The port width is automatically set to 95% of the wire width and the rotation is automatically set so that the port connects across the width of the wire. When you change the port rotation for the *Cross layer Type*, the rotation value for positive connection of the port is updated synchronously with the change in the rotation value for negative connection of the same port and vice-versa.

The *Components* sub-tab lists the instances connected to the nets you selected for a model.

© 2023 Cadence Design Systems, Inc.



4. (Optional) On the *Ports* sub-tab, click (*Add Port*) to create a new port.

   For each port, two rows are added in the table below the ports table. These rows show the details of connections of the new port. By default, the X and Y coordinates of these connections are set to 0.

5. (Optional) Select a row in the ports table and click *Move Connection* at the bottom to move the coordinates. You can also modify the layer name for the edges.

6. Add the pin name and instance name for the newly added port.

   Note: When using the *Add Port* command to manually create a new port, it is mandatory to provide the pin name if you plan to create an extracted view after running an EM simulation.

7. (Optional) If you do not want any port that was added, select its row and click (*Remove Port*) to delete that port.

8. Use the following commands on the *Setup* tab to manage the ports:

   - Click (*Highlight All Ports in the Layout Canvas*) to view the ports highlighted on the layout canvas.
   - Use the (up and down) arrows to move the rows in the ports table.

9. On the *Components* tab, review and configure the properties of the ports to be created depending on their instance type.

© 2023 Cadence Design Systems, Inc.

Port setup for the Clarity model is complete. You can check schematic binding or validate ports.

Related Topics

Ports Tab (Clarity with local reference)

Validating Ports Created for Clarity Models

Checking Schematic Bindings for Ports Created for EMX Models

© 2023 Cadence Design Systems, Inc.

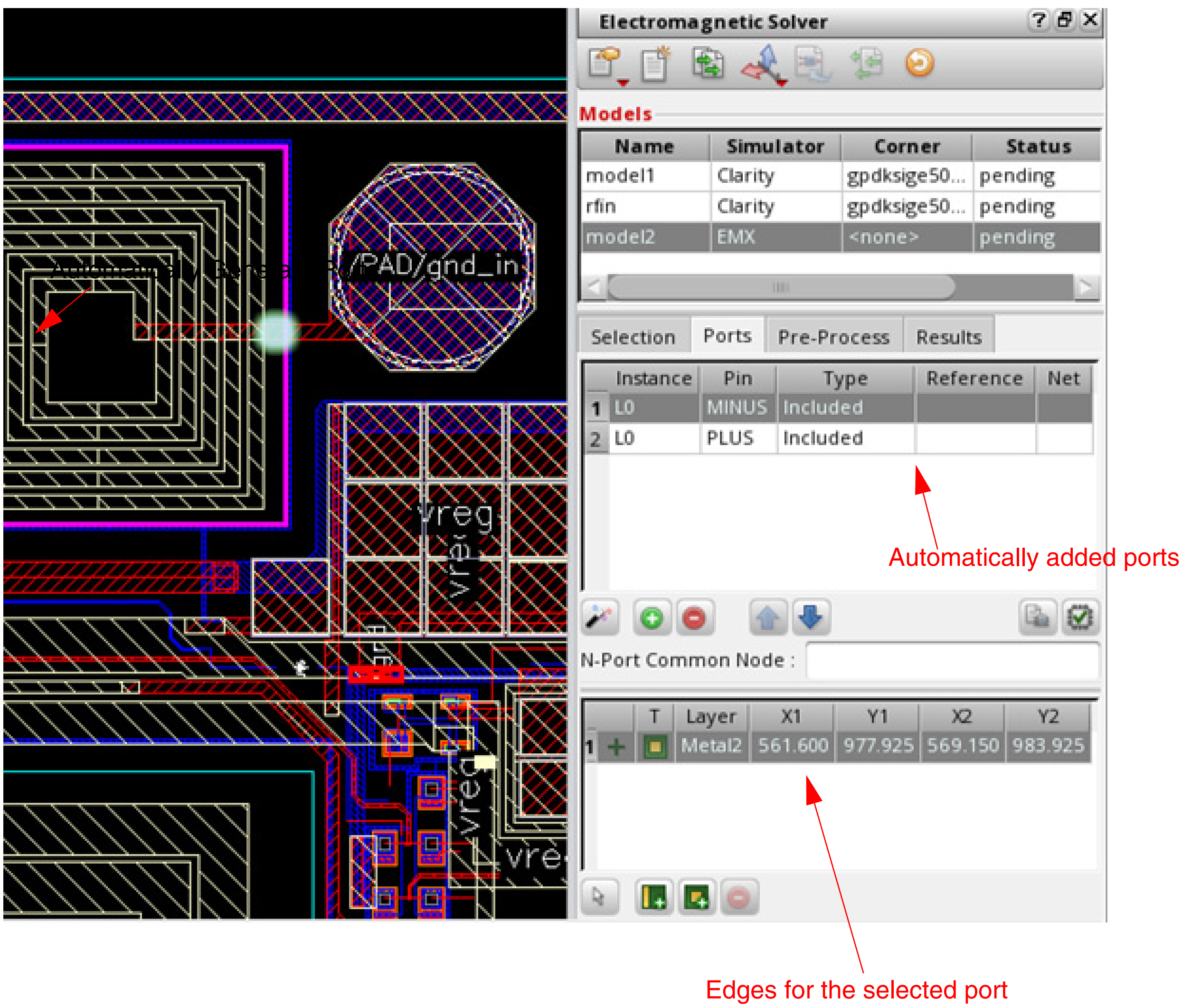
Specifying Ports for EMX Models in IC Layouts

For the models that use EMX simulator, you can begin port creation with the automatic port generation feature to create ports for the selected objects. After that, you can modify the ports created by the tool or manually add more ports.

To add ports to a model:

1. In the *Ports* tab, click *Automatically Generate Ports* to generate ports for the instances selected on the *Selection* tab.

   The tool uses certain rules to automatically identify pins and creates ports and displays the port details in the ports table. The table below the ports table shows the edges created for each port, as shown below.



© 2023 Cadence Design Systems, Inc.

2. To create a new port, click *Add Port* ( ) in the *Ports* tab.

   A new row is added in the ports table.

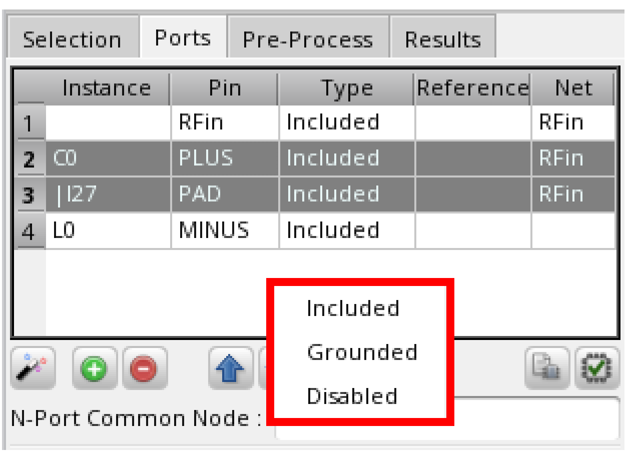
3. In the *Instance* column of the new row, specify the name of an instance to which the port is connected.

4. In the *Pin* column, specify the name of the terminal to which the port is connected.

5. In the *Type* column, choose a port type from the drop-down list. A different list of port types is shown for each simulator.

   Note: Disabled and grounded ports do not have an index number because these are not used for simulation.

   For models using EMX, you can set a common port type for multiple ports. Select the rows for multiple ports, right-click, and choose *Set Port Type*. A list showing the available port types is displayed at the pointer location. Choose a port type to be used. The selected port type is set for the selected rows.



6. (For EMX models only) In the *Reference* column, specify the name of the port to be used as the reference for this port. The port is assigned as the `Disabled` port type automatically.

   To set a common reference port for multiple ports, select the rows for those ports, right-click, and choose *Set Reference Port*. In the form that is displayed, specify the name of the pin and click *OK*.

   Note: If you specify pin that is not a disabled port, the cell is highlighted in red. Correct the reference port name for such cases.

7. Similarly, you can add more ports.

© 2023 Cadence Design Systems, Inc.

8. Specify a net name in the *N-Port Common Node* field. The reference pin of the N-port instance in the schematic is connected to this net. If not specified, this field uses the first ground net found in the model.

   Note: If the setup contains an N-port common reference net, the cellview may not be compatible with an earlier release that does not support this feature.

While working with the ports, you can use the following commands in the *Ports* tab:

- *Remove Port* ( ): Removes the selected port. To remove all ports, right-click any row, choose *Select All*, and then use this command.
- Up and down arrows ( ): Move the selected ports up or down. Use these commands to keep the ports of interest in focus.

Related Topics

- Rules for Port Generation for EMX Models in IC Layouts
- Port Management for EMX Models in IC Layouts

© 2023 Cadence Design Systems, Inc.

Port Management for EMX Models in IC Layouts

Meaningful electromagnetic simulation results depend greatly on the quality of port definitions. If the port definitions are not correct for your layout, you may get results, but many times, they cannot be trusted. Therefore, it is important to validate them and make modifications, where required.

The *Ports* tab of the Electromagnetic Solver assistant provides various commands to check the ports, change the default internal ports created automatically for EMX models to edge ports, and to change the placement or alignment of the ports.

Related Topics

Managing Edges for Ports Created for EMX Models

Checking the Validity of Ports Created for EMX Models

Checking Schematic Bindings for Ports Created for EMX Models

Blog: Virtuoso Meets Maxwell: Defining Ports in EMX Planar 3D Solver

© 2023 Cadence Design Systems, Inc.

Managing Edges for Ports Created for EMX Models

For EMX models, the table at the bottom of the *Ports* tab shows the edges created for each port. In this table, you can manage edges for ports or create internal ports.

To create an internal port for a pin:

1. Click *Add Port* to create a new port in the ports table.

   A blank row is added to the ports table. A blank row with default X and Y coordinates is added to the edges table.

2. Specify a pin name and net name in the new row that was added in the ports table.
3. Click *Add Internal Connection* ( ) at the bottom of the *Ports* tab.

   The pointer is changed to indicate that a command for port creation is in progress.

4. Click the pin for which you want to create an internal port.

   The tool creates a port at the pin location. The size of the port is same as that of the pin.

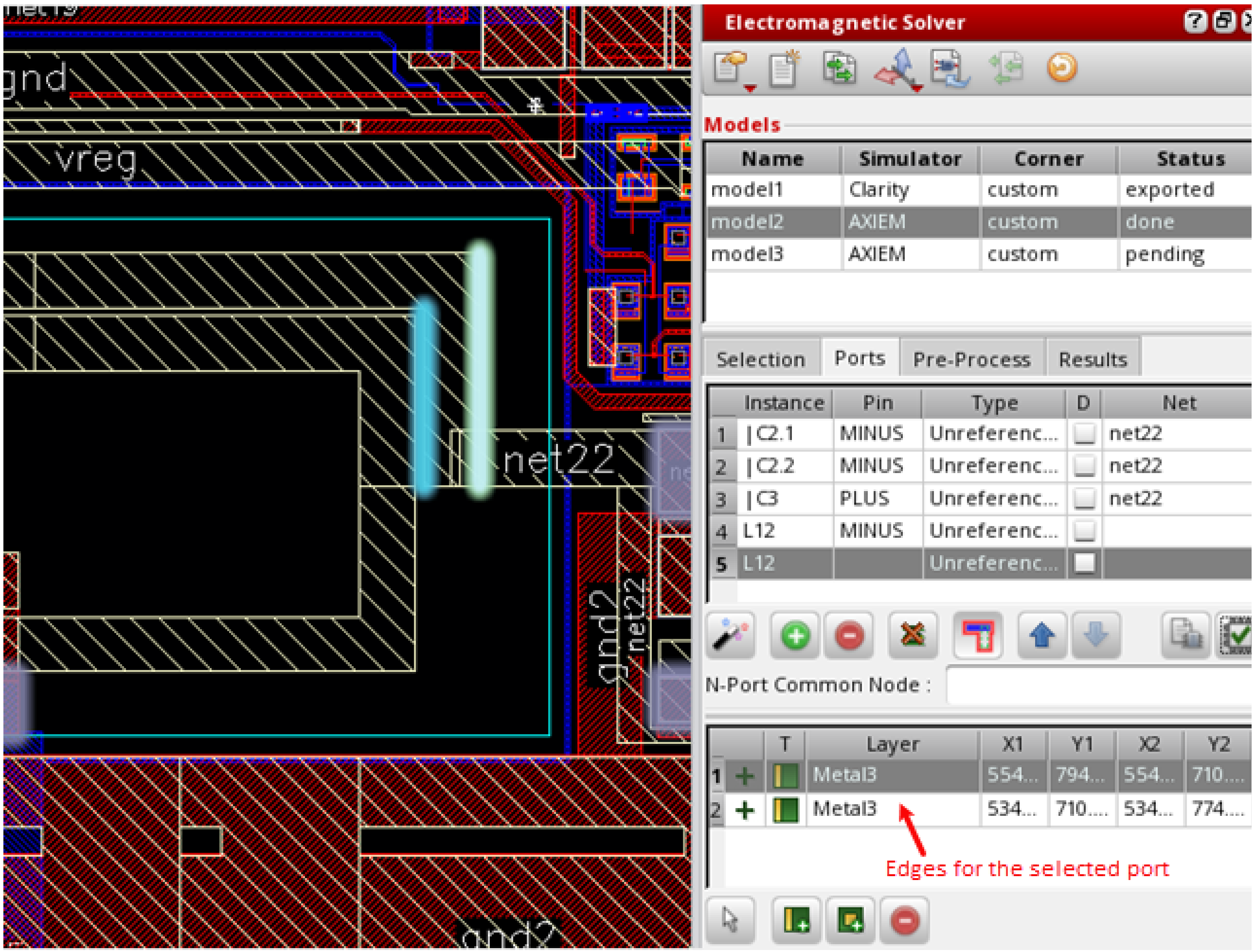
5. Select the undesired row in the edges table and click *Remove Connection* to delete the row.

To complete the edge details for ports:

1. Click *Add Edge Connection* ( ) at the bottom of the *Ports* tab.
2. Click an edge on a selected shape in the model.

© 2023 Cadence Design Systems, Inc.

An edge is added and highlighted on the layout canvas. The layer name and coordinates of the edge are also shown in the edges table.



3. (Optional) Use the *Move Connection* ( ) command to change the location of an edge.
4. Verify the connections by using the *Validate Ports* command. For details, see **Checking the Validity of Ports Created for EMX Models**.

Proceed with the generation of mesh and port connection review.

© 2023 Cadence Design Systems, Inc.

Checking the Validity of Ports Created for EMX Models

To validate the ports:

➡ Click *Validate Ports* on the *Ports* tab.

The assistant checks each port and displays errors for invalid ports.

For EMX, the tool checks that a port should not have coordinates 0,0. For an internal port, the tool exports a shape with the size of that port is exported to the solver. Therefore, by default, an internal port is always considered to be valid. You can manually check the shapes corresponding to internal ports in gdsview.

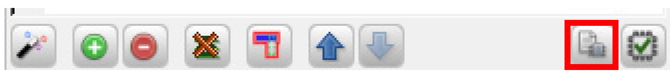
© 2023 Cadence Design Systems, Inc.

Checking Schematic Bindings for Ports Created for EMX Models

After port generation, you can check the schematic binding and resolve any XL binding issues before running a simulation.

To check the schematic binding:

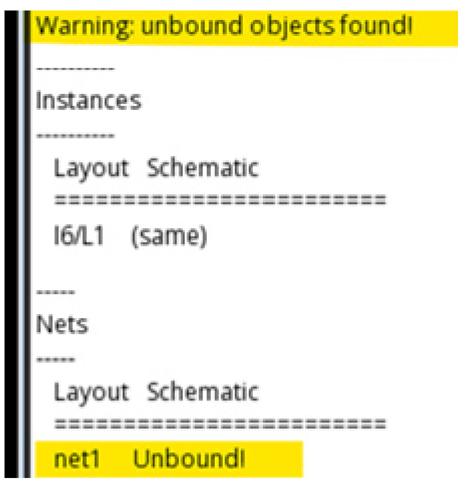
➡ Click *Report Schematic Binding* on the *Ports* tab.



Note: This command is not visible when *Extract Full Cellview* is selected on the *Selection* tab.

The tool displays a binding report in ViewFile window.

In the following example, the report shows an unbound net in the model.



```
Warning: unbound objects found!
----------
Instances
----------
  Layout  Schematic
  ========================
  I6/L1   (same)

-----
Nets
-----
  Layout  Schematic
  ========================
  net1    Unbound!
```

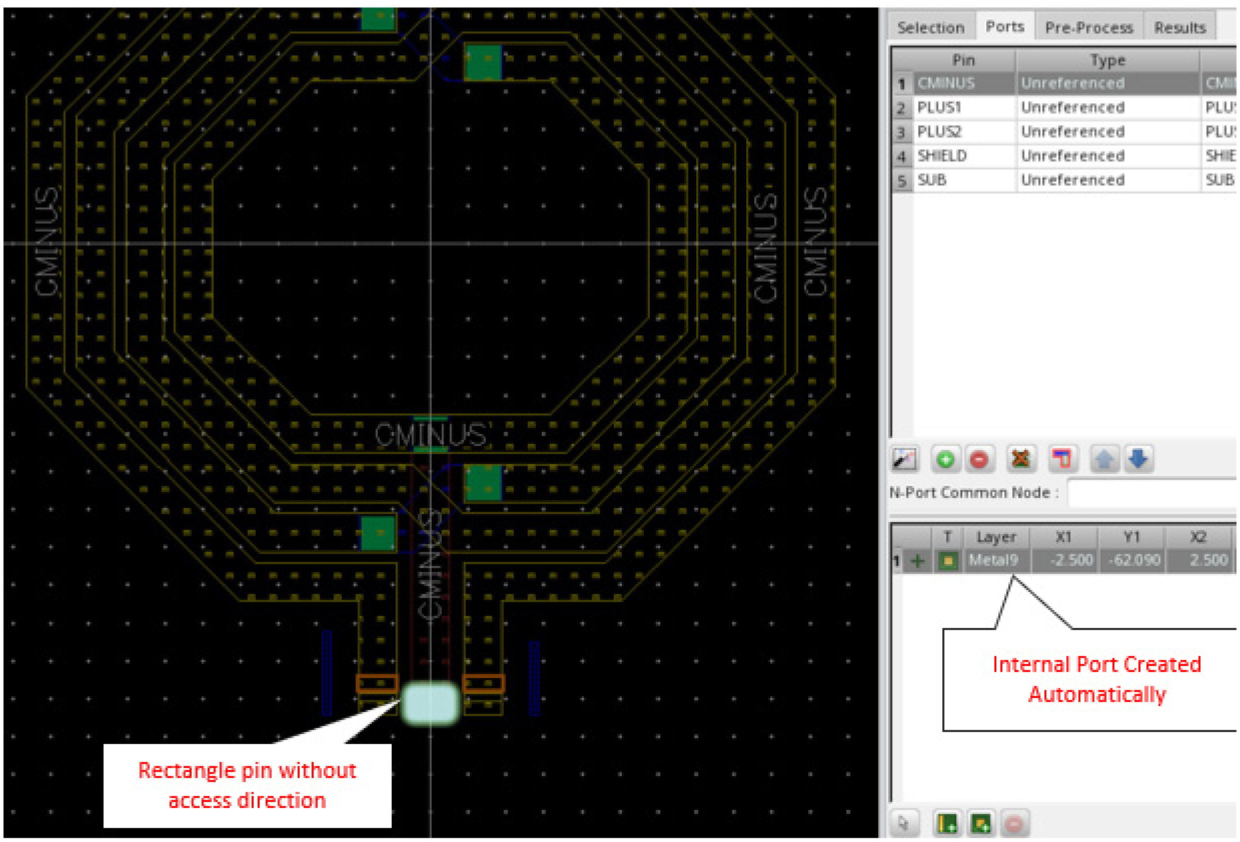
Resolve such errors before model generation because these can cause issues while creating extracted views.

© 2023 Cadence Design Systems, Inc.

Rules for Port Generation for EMX Models in IC Layouts

When you use the *Automatically Generate Ports* command in the *Ports* tab to automatically generate ports for the instances selected on the *Selection* tab, the tool uses the following rules to generate ports for an EMX model:

- For top-level pins:
  - Creates ports at top-level pin locations.
  - If a pin has an access direction, creates an edge port in that direction.
  - If a pin does not have any access direction, creates an internal port that is the same size as the pin.

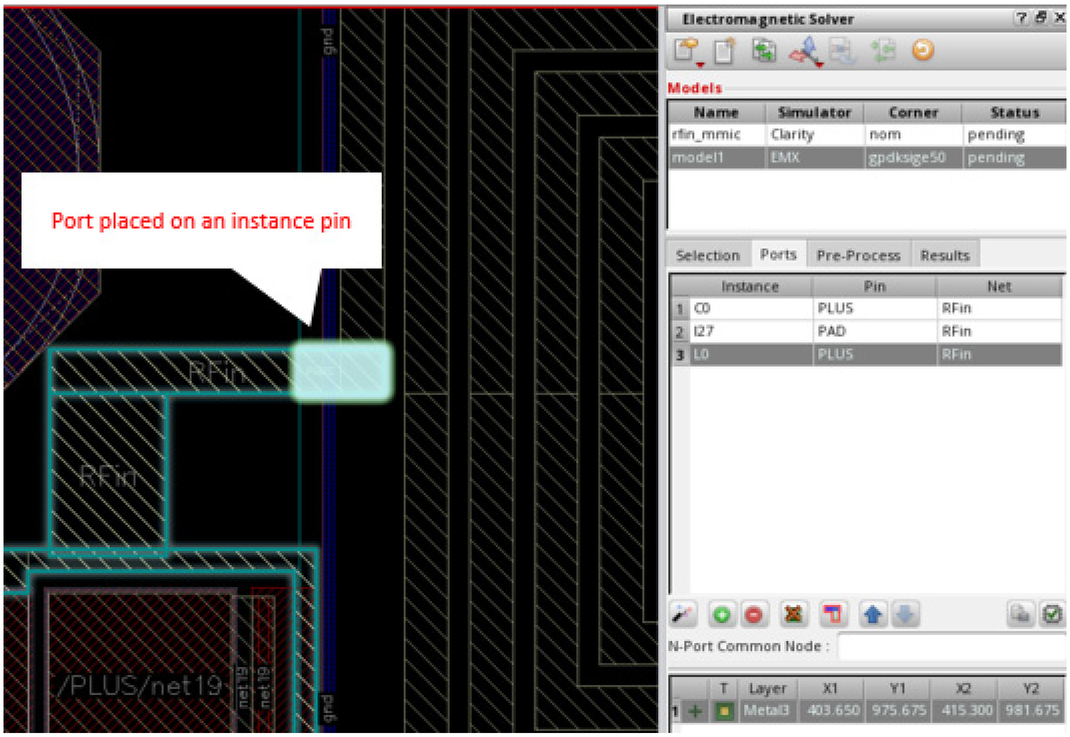


To create an edge port for a pin that has an internal port, specify an access direction for that pin.

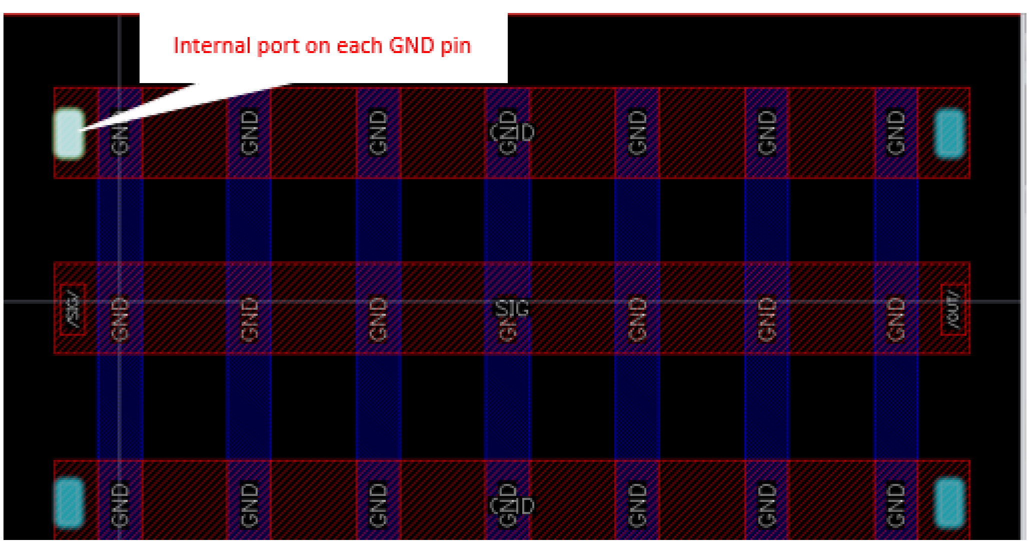
To specify the access direction, right-click the pin and choose *Properties* to open the Edit Rectangle Pin Properties form. Select the appropriate check box to specify the access direction you require and apply the changes. Next, rerun automatic port generation to create an edge port for the updated pin.

© 2023 Cadence Design Systems, Inc.

- For a net connected to an instance that is not in the model, creates a port at the instance pin to which that net connects.



- If multiple pins with same name are present, the tool creates an internal port on each pin. EMX considers such ports are electrically connected.



- For pins with multiple pin figures, creates a port using the largest of the available pin figures.

© 2023 Cadence Design Systems, Inc.

- If two pins touch or overlap, they share the same port.
- For instances that have terminals with the connection status set to `mustJoin`, combines all must-join fingers of a terminal into a single port. For example, for a MOS device that has multiple fingers with the `mustJoin` property for the G, S, or D terminals, the tool creates only three ports for the device. One port is created for each MOS pin regardless of the number of fingers.
- For layout instances that correspond to a single schematic instance with a multiplier, a port is created for each schematic pin and a connection to the port is added for each layout pin. For example, if a MIMCAP instance has a multiplier m=2 in the schematic, then the tool creates two ports for the MIMCAP. The port for the PLUS terminal would be connected to the PLUS pin of both MIMCAP layout instances.

  Note: This does not apply when *Extract Full Cellview* is enabled.

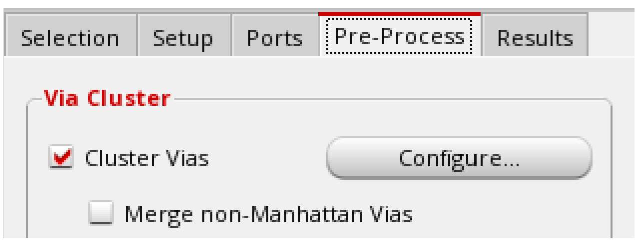
© 2023 Cadence Design Systems, Inc.

Specifying Options for Via Clustering

Via clustering requires the maximum spacing and maximum number of cuts for each via layer. Specifying options for via clustering involves configuring these settings and enabling via clustering.

To specify options for via clustering:

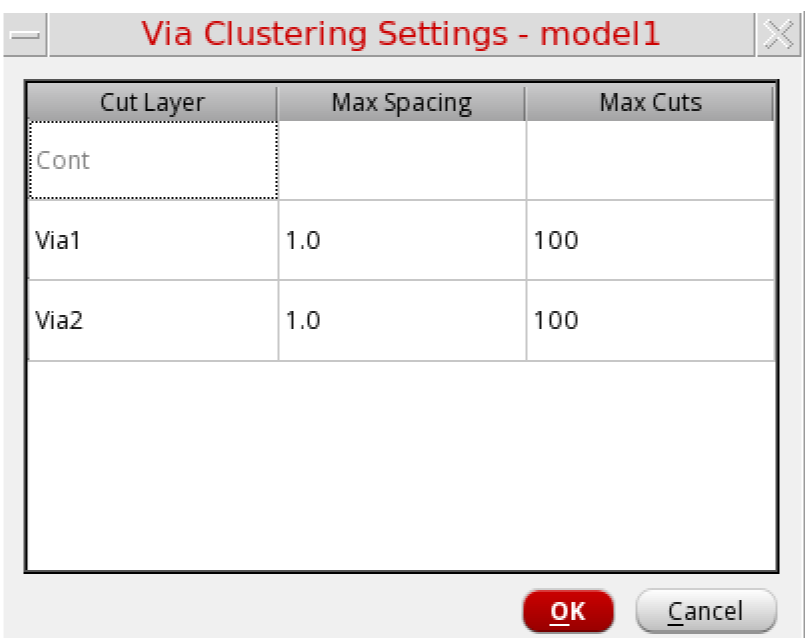
1. Select a model in the Models table of the Electromagnetic Solver assistant.
2. On the *Pre-Process* tab of the assistant, select the *Cluster Vias* check box in the *Via Cluster* group.



   This step enables via clustering for your model.

3. Click *Configure* to open the Via Clustering Settings - `model-name` form.

   The Via Clustering Settings form is displayed.



© 2023 Cadence Design Systems, Inc.

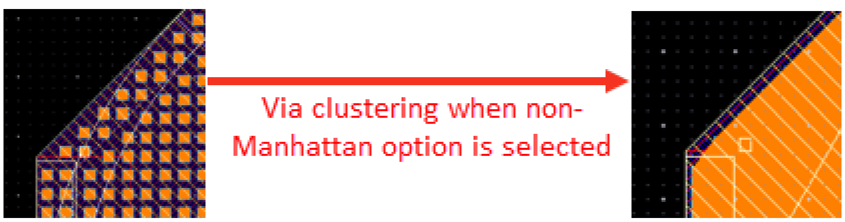
4. Review the settings displayed in the table. It shows the default values for maximum spacing and maximum cuts that the tool read from the `default.viacl` file while creating a new model. For the cut layer names not found in the `default.viacl` file, it leaves the cells blank.

5. If you need to use a different configuration for a specific model, change the imported values.

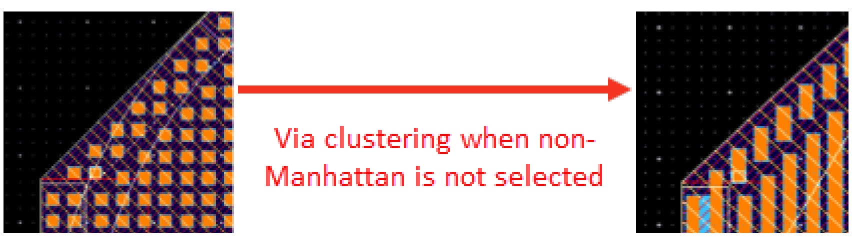
6. Add details for the blank cells, if any.

7. Click *OK* to close the Via Clustering Settings form.

8. If your layout contains vias that are not aligned in orthogonal rectangles, select *Merge non-Manhattan Vias* in the *Via Cluster* group on the *Pre-Process* tab to cluster all vias. This setting ignores the *Max Cuts* via cluster setting and clusters the maximum possible vias.



When this option is not selected, vias that are not aligned as rectangles are not clustered.



Saving Default Parameters for Via Clustering

You can specify the default settings for via clustering in the `default.viacl` file saved at the directory location specified by the `processCornerDirectory` environment variable.

© 2023 Cadence Design Systems, Inc.

The following example shows the format in which values are specified in the `default.viacl` file:

```
(viaClusterParams
     ("Via2" maxCuts 100 maxSpacing 1.0)
     ("Via1" maxCuts 100 maxSpacing 1.0)
)
```

Any new model created in the Electromagnetic Solver assistant reads the default settings from this file and populates the values in the Via Clustering Settings - *<model-name>* form.

Related Topics

Pre-Process Tab

processCornerDirectory

© 2023 Cadence Design Systems, Inc.

Specifying Shape Simplification Options for IC Layouts

After selecting shapes for inclusion in a model, you can specify shape simplification options that can be considered by the tool while preparing the model for submission to the electromagnetic solver. These options are used to reduce fine details in those areas of the layout that might not impact the result of electromagnetic simulation. For example, you can remove small dangling shapes or merge closely placed shapes. Such changes reduce the number of shapes to be considered during the simulation run and improves the speed of the electromagnetic simulation run.

Important

Shape simplification works only on connected shapes. Therefore, it is essential to have a connectivity-driven layout to run shape simplification.

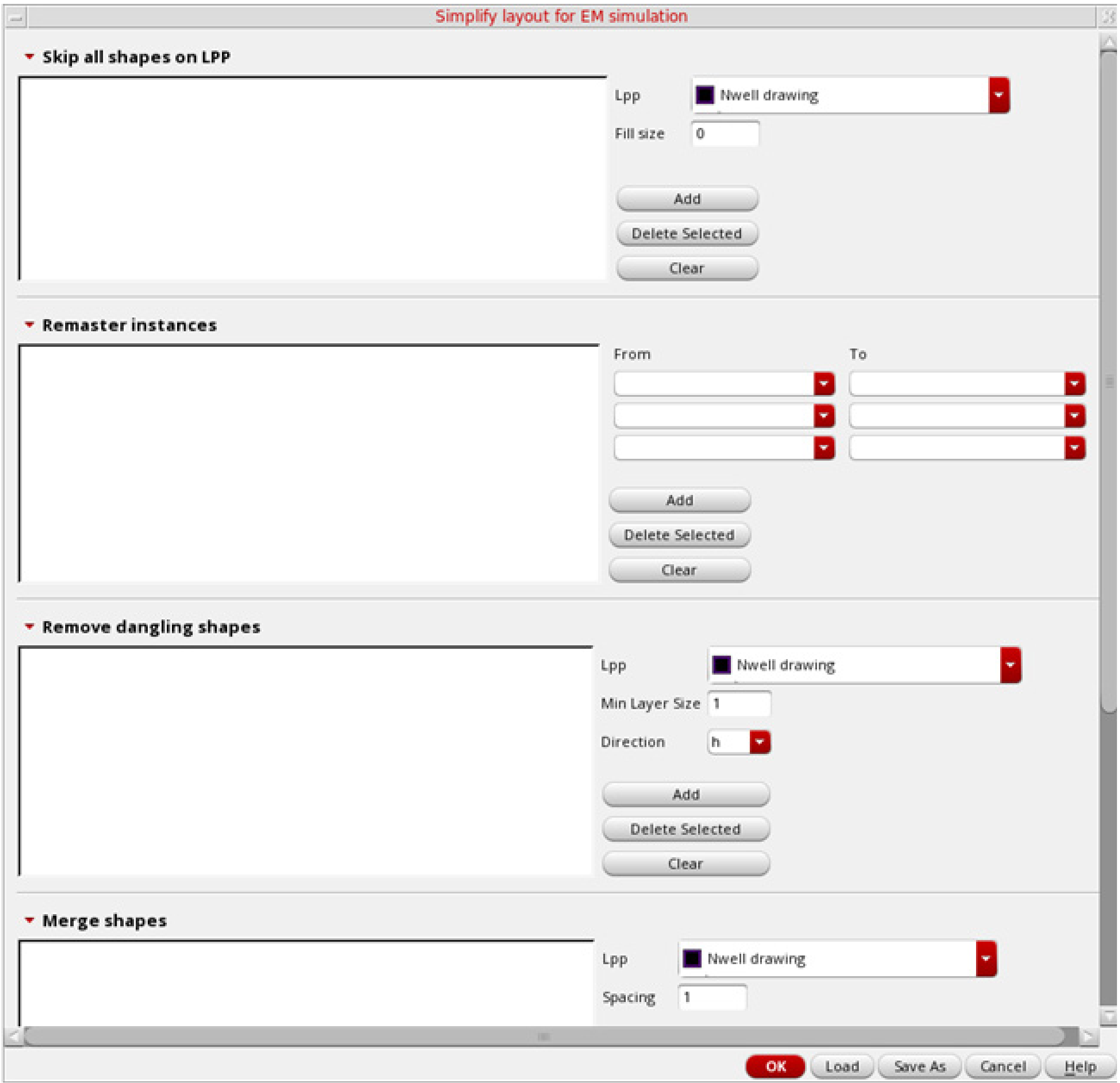
Using the setup on the *Pre-Process* tab, you can configure a shape simplification setup with details of the shapes and the simplification activities to be performed for them. These settings can be saved in a recipe file, which is by default saved in the current working directory. A recipe file can be loaded later to reuse the settings.

To specify the settings for shape simplification:

1. Click *Setup* on the *Pre-Process* tab of the Electromagnetic Solver assistant.

© 2023 Cadence Design Systems, Inc.

The Simplify layout for EM simulation form is displayed.



This form contains six sections in which you can specify options to simplify shapes according to your layout and shapes selected in the model.

Related Topics

Specifying Options to Skip Shapes on Selected LPPs

© 2023 Cadence Design Systems, Inc.

Specifying Options to Change Instance Masters

Specifying Options to Remove Dangling Shapes

Specifying Options to Merge Shapes

Specifying Options to Smooth Shape Steps

Specifying Options to Skip Shapes on Selected LPPs

Certain processes require dummy metal fills to meet certain density requirements, but you may want to run simulations for those processes by eliminating the metal fill. To eliminate or skip metal fills based on their fill size, add rules in the *Skip all shapes on LPP* section of the Simplify layout for EM simulation form.

To add a rule in the *Skip all shapes on LPP* section:

1. Choose the name of a layer-purpose pair from the *Lpp* drop-down list.
2. Specify the minimum fill shape size in the *Fill Size* field.

   For example, if the metal fill consists of 3 micron by 3 micron squares, enter `3` in *Fill size*.

   Note: To skip the complete layer-purpose pair, enter `0` in the *Fill size* field.

   All shapes that are less than the specified size are skipped while running a simulation.
3. Click *Add* to add a new rule for the specified layer in the list.

   You can use the *Delete Selected*, or *Clear* commands to add delete a rule or to delete all rules from the list.

   Note: While eliminating the shapes based on their fill size, the tool does not need to extract the layout. This means that shape simplification is faster than removing dangling shapes, which requires extraction. However, there is an exception for the models with simulator set to `Clarity`. The layout is always extracted for these models because the tool needs to derive the net names to be shown in Clarity 3D window.

Related Topics

Specifying Shape Simplification Options for IC Layouts

Specifying Options to Change Instance Masters

Specifying Options to Remove Dangling Shapes

© 2023 Cadence Design Systems, Inc.

Specifying Options to Merge Shapes

Specifying Options to Smooth Shape Steps

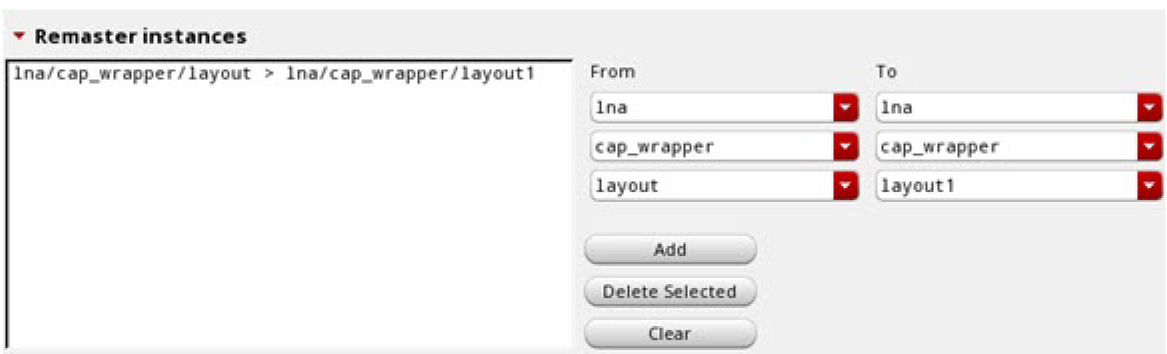
Specifying Options to Change Instance Masters

If you want to modify the master layout cellview for all instances of a particular layout cellview during the simulation run, you can specify rules to remaster instances in the *Remaster instances* section of the Simplify layout for EM simulation form.

To add a new rule to remaster instances:

1. Choose the library, cell, and view name of the existing layout from the drop-down lists given below *From*.
2. Choose the library, cell, and view name of the new master layout from the drop-down lists given below *To*.
3. Click *Add*.

   A new simplification rule is added to the list in the *Remaster instances* section. An example is shown below.



Related Topics

Specifying Shape Simplification Options for IC Layouts

Specifying Options to Skip Shapes on Selected LPPs

Specifying Options to Remove Dangling Shapes

Specifying Options to Merge Shapes

Specifying Options to Smooth Shape Steps

© 2023 Cadence Design Systems, Inc.

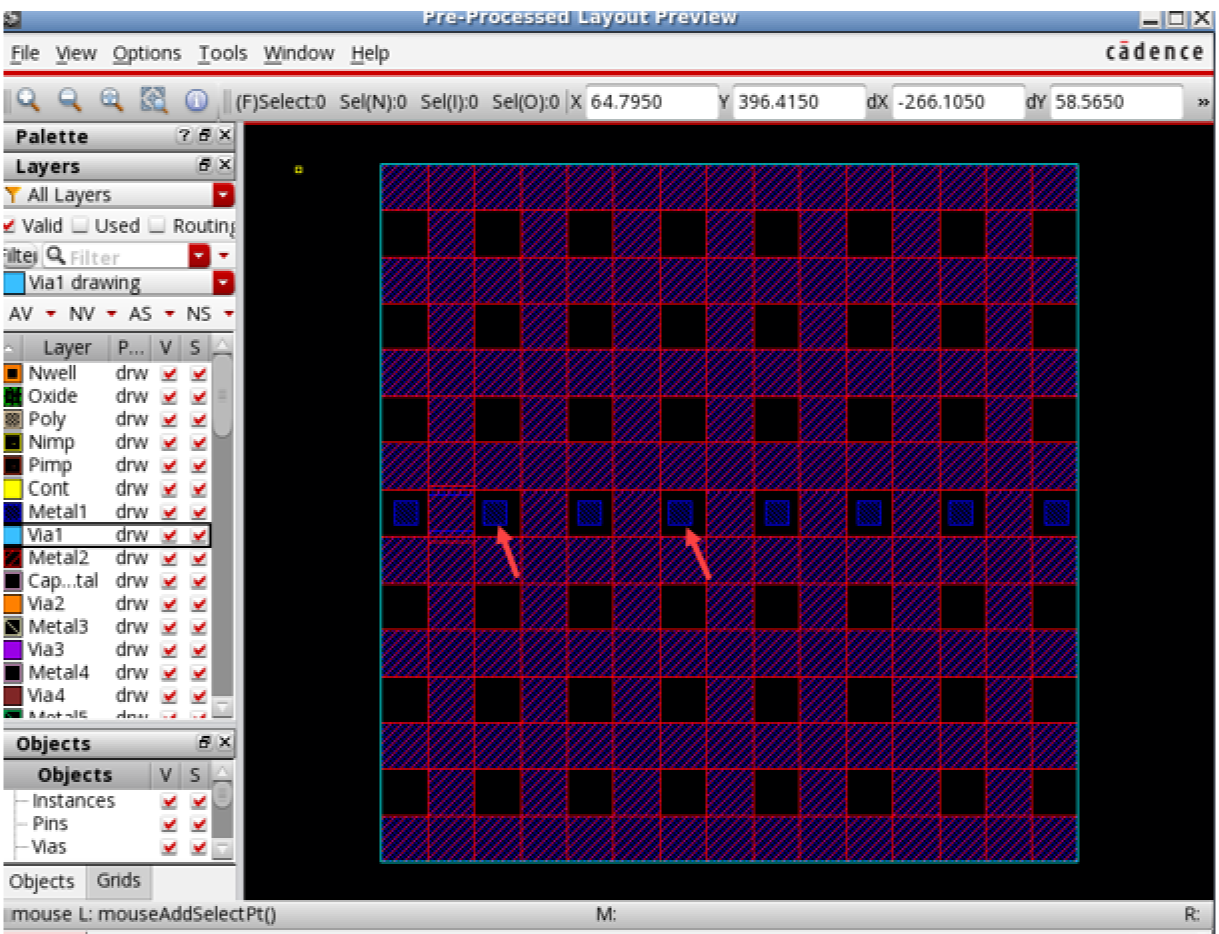
Specifying Options to Remove Dangling Shapes

If the layout contains dangling shapes, you can specify rules in the *Remove dangling shapes* section of the Simplify layout for EM simulation form to remove those before sending the model to the simulator.

To add a new rule to remove dangling shapes based on their size from one or more layer-purpose pairs in the layout:

1. Select a layer-purpose pair from the *Lpp* drop-down list.
2. In the *Min Layer Size* field, specify the minimum size of shapes you want to retain on the layer. The tool removes any shape smaller than the specified size.

   For example, if there are floating `metal1` shapes smaller than `10x10` microns, you can create a rule for the `metal1` drawing layer-purpose pair to remove all shapes less than 10 microns, as pointed by the red arrows in the following example of a pre-processed layout.



© 2023 Cadence Design Systems, Inc.

3. Click *Add* to create and add the rule to the list in this section.

Related Topics

Specifying Shape Simplification Options for IC Layouts

Specifying Options to Change Instance Masters

Specifying Options to Skip Shapes on Selected LPPs

Specifying Options to Merge Shapes

Specifying Options to Smooth Shape Steps

Specifying Options to Merge Shapes

If the layout contains various shapes on the same layer or on the layers above or below a specific LPP, you can specify rules to merge shapes if they are within the given distance.

To add a new rule in the *Merge shapes* section of the Simplify layout for EM simulation form:

1. Select a layer-purpose pair on which you want to merge shapes in the *Lpp* drop-down list.
2. Specify the minimum allowed distance between the devices in the *Spacing* field.
3. Click *Add*.

A rule is added to the list in this section. You can create multiple rules like these for different layer-purpose pairs. If the layers are closer than the minimum allowed distance, they are merged.

For example, the following rule set will merge `metal1` and `metal2` if they are within `20 microns`.

© 2023 Cadence Design Systems, Inc.

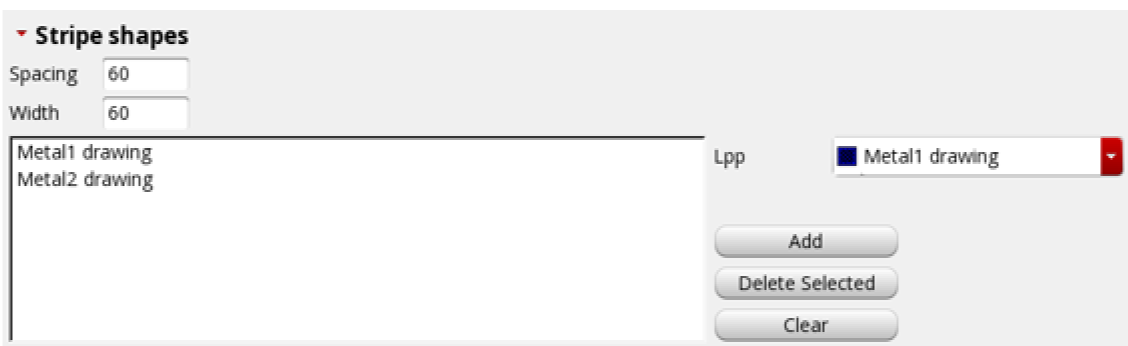
Specifying Options to Stripe shapes

By creating rules to convert layers into stripes of the given width, you can convert fine mesh into a coarse mesh that would contain a smaller number of unknowns to be passed to the solver for simulation.

To add a new rule in the *Stripe shapes* section of the Simplify layout for EM simulation form:

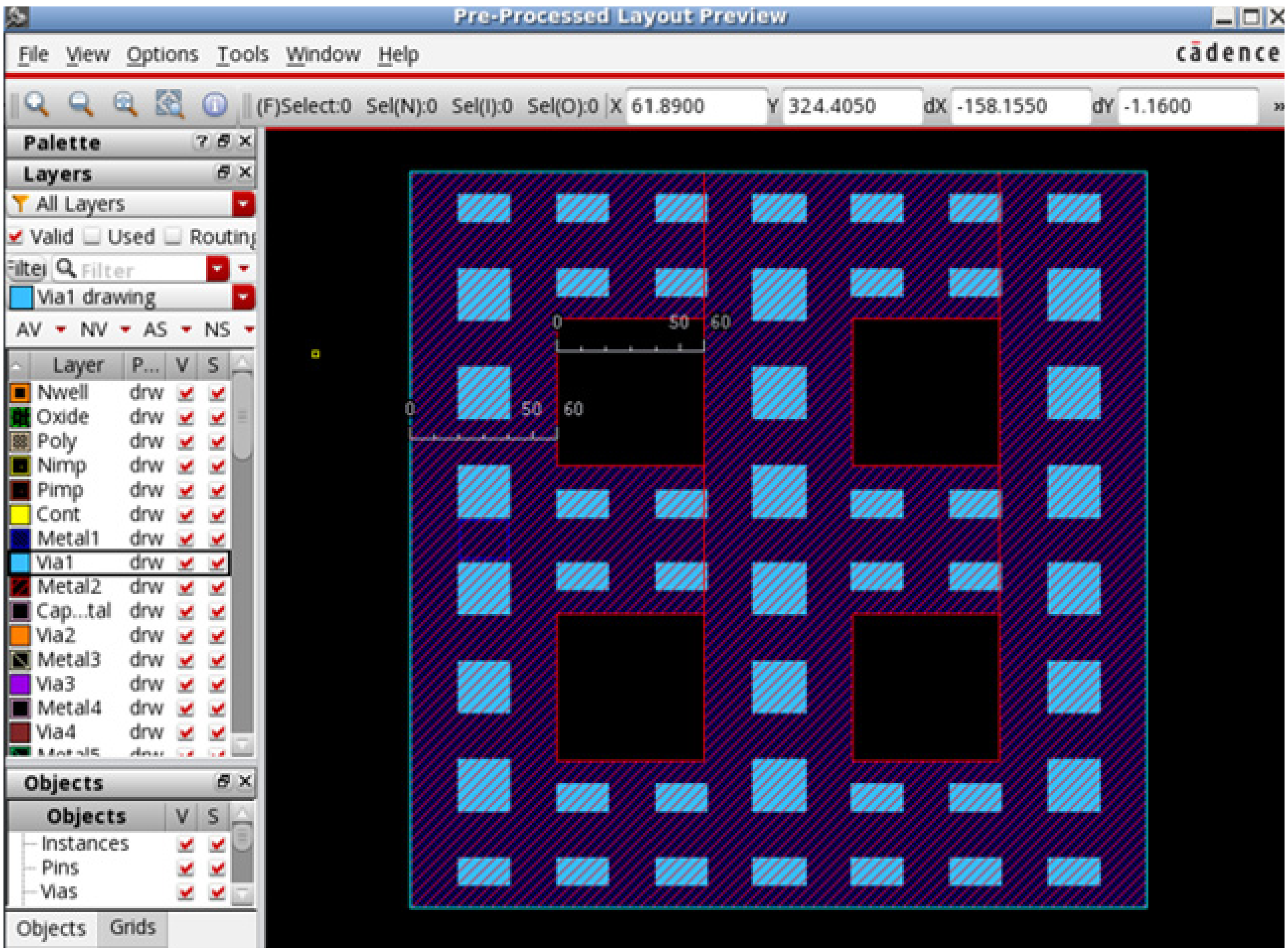
1. In the *Lpp* drop-down list of this section, select a layer-purpose pair that you want to convert into stripes.
2. Specify the width for stripes in the *Width* field.
3. Specify the spacing to kept between each stripe in the *Spacing* field.
4. Click *Add*.
5. (Optional) You can also define a via enclosure value in the *Via enclosure* field. This value specifies the distance by which the vias must be pulled in from the metal edge.

   The following example shows a set of rules to create stripes for metal1 and metal2. The stripes are 60 microns in size and placed 60 microns apart.



© 2023 Cadence Design Systems, Inc.

The preview of the pre-processed layout given below shows the stripes created according to the rules mentioned in the example given above. Note the measurements.



Related Topics

Specifying Shape Simplification Options for IC Layouts

Specifying Options to Change Instance Masters

Specifying Options to Remove Dangling Shapes

Specifying Options to Skip Shapes on Selected LPPs

Specifying Options to Smooth Shape Steps

© 2023 Cadence Design Systems, Inc.

Specifying Options to Smooth Shape Steps

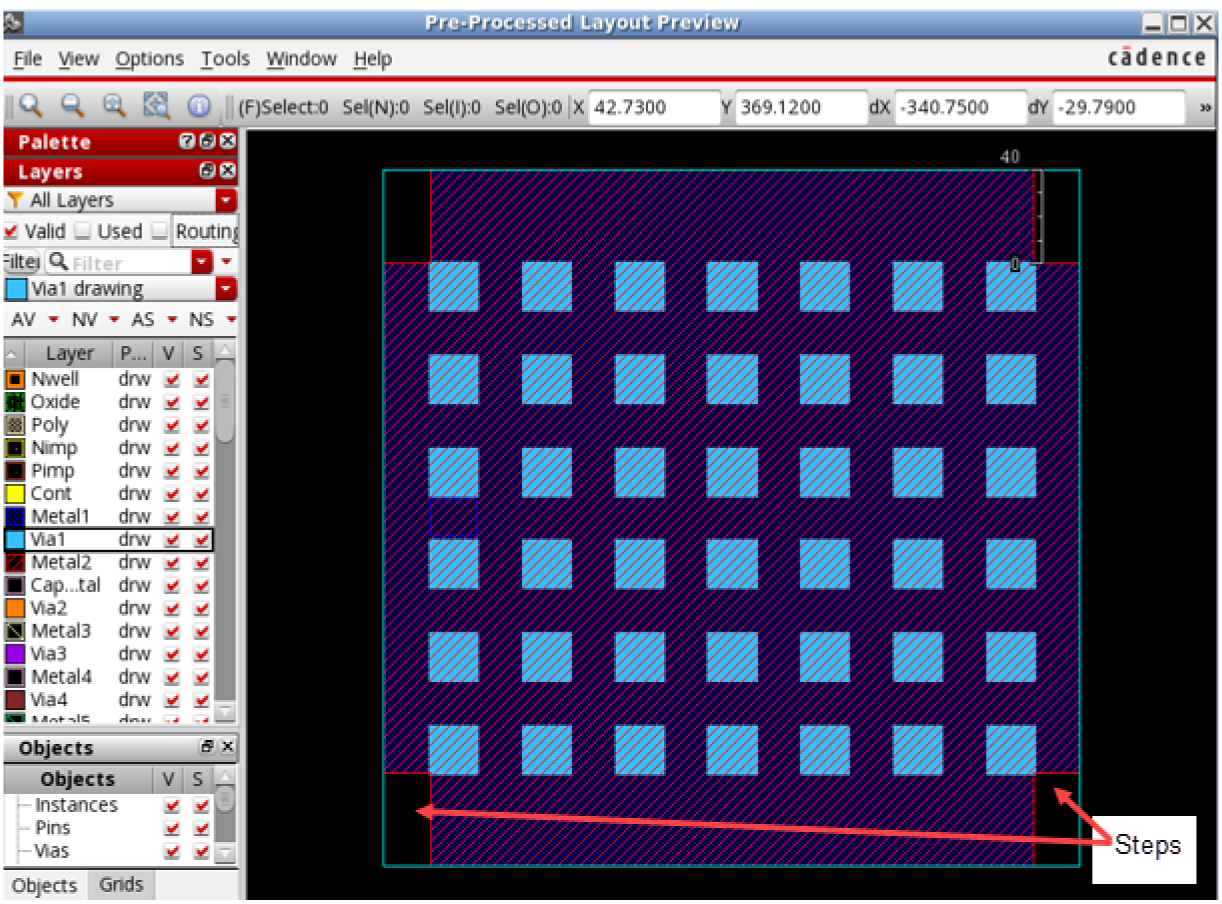
If you created rules to stripe the shapes, you can smoothen out small steps or edges that remain after that. This also helps in reducing the number of unknowns in the layout.

To add a new rule to smooth a shape:

1. From the *Lpp* drop-down list, select the layer-purpose pair for the shapes to be considered.
2. Specify the minimum allowed step size in the *Step Size* field.
3. Click *Add* to create a rule.

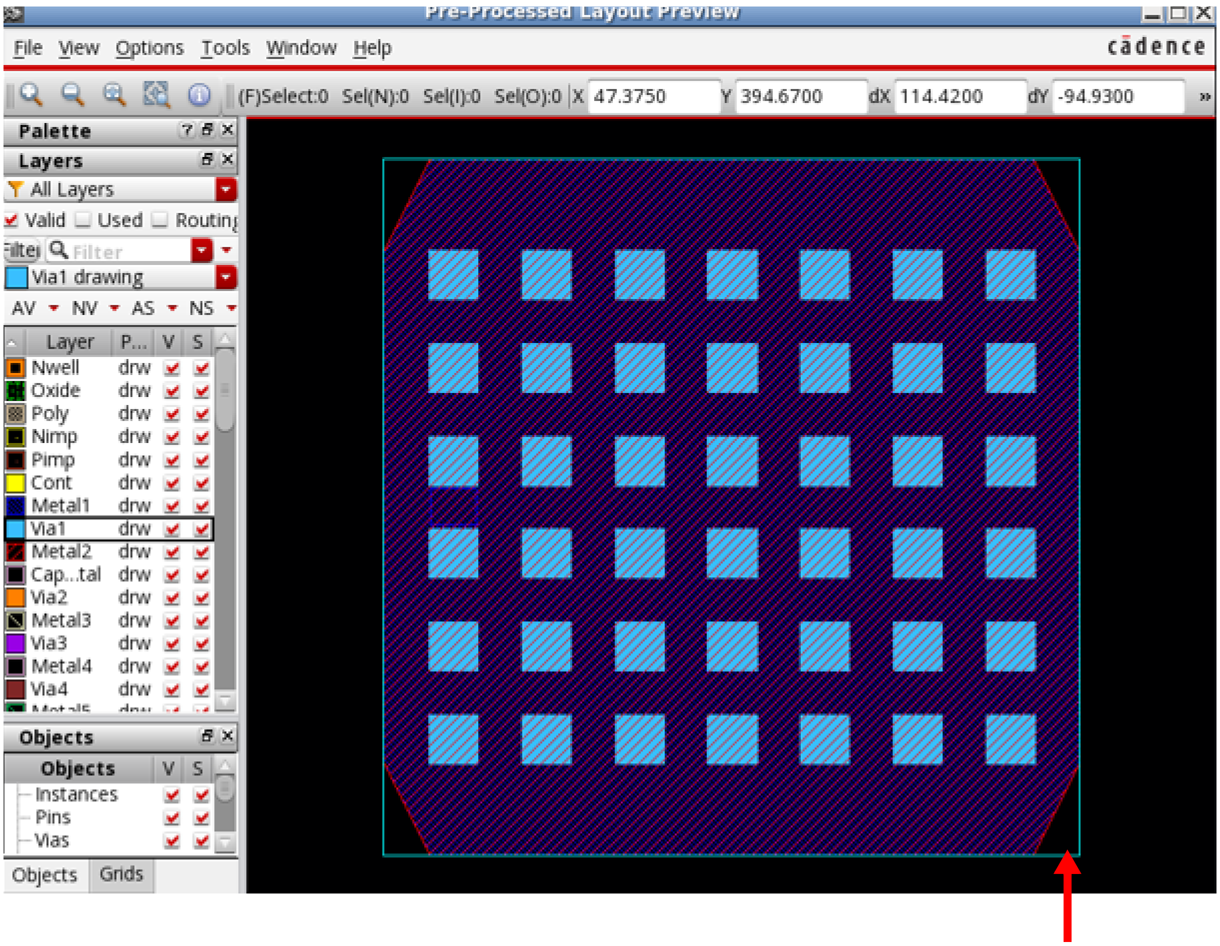
   All the steps with size less than the given limit are smoothened.

Consider the following example of a layout. After the shapes on a metal layer are merged, there are steps remaining in all four corners.



© 2023 Cadence Design Systems, Inc.

You can measure the size of steps and add a rule to smoothen them out. After smoothing, the example layout appears as shown below.



After smoothing

Related Topics

Protecting Layout Objects from Shape Simplification

Previewing Pre-Processed Layout

Simplify Layout for EM Simulation Form

Pre-Process Tab

© 2023 Cadence Design Systems, Inc.

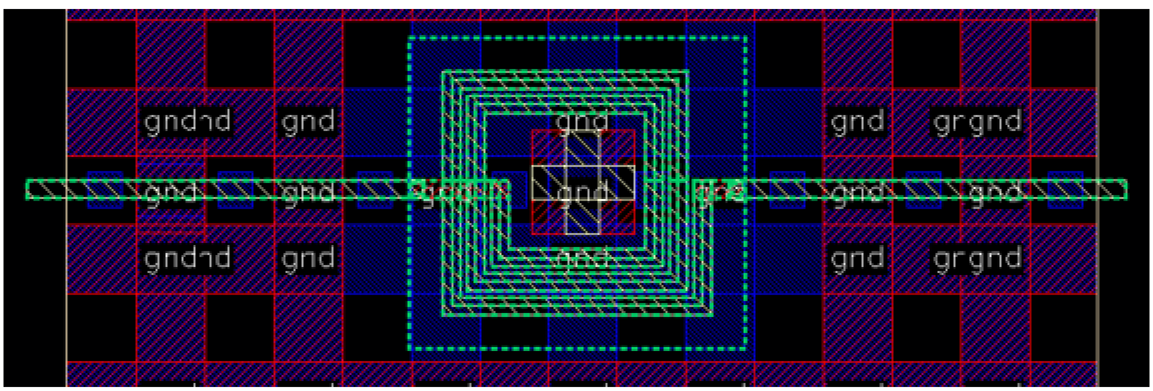
Protecting Layout Objects from Shape Simplification

All the modifications done to simplify shapes are done based on the rules specified in the Simplify Layout for EM Simulation form. The main objective of shape simplification is to reduce fine details in those areas of the layout that might not impact the result of electromagnetic simulation. For example, you can remove small dangling shapes or merge closely placed shapes. Such changes reduce the number of shapes to be considered during the simulation run, making the process faster. However, if there are certain important shapes in your design that should be used in simulation, you can protect those from processing.

If you want to protect certain shapes from modifications for shape simplification even if they meet the rules:

1. Select the desired shape on the layout canvas.

   For easy identification of the protected shapes, select the *Highlight Protected* check box. The protected shapes are highlighted on the layout canvas in green, as shown below.



2. Click *Protect Selected* on the *Pre-Process* tab of the Electromagnetic Solver assistant.

   The tool ignores the selected shapes while running shape simplification.

Guidelines for Protecting Shapes

You can consider the following guidelines while selecting shapes for protection:

- The shapes at any level of design hierarchy can be protected.
- The shape protection information is saved in the layout cellview. Therefore, after you protect one or more shapes, the layout cellview is marked as modified.

© 2023 Cadence Design Systems, Inc.

- The same set of protected shapes is applicable for all the models defined for a layout view.

Related Topics

Specifying Shape Simplification Options for IC Layouts

Previewing Pre-Processed Layout

Pre-Process Tab

© 2023 Cadence Design Systems, Inc.

Previewing Pre-Processed Layout

Before submitting a model to the EM simulator, you can check its preview to verify the shapes and geometries included in it.

To check the preview:

1. Select a model in the *Models* table of the Electromagnetic Solver assistant.
2. On the Pre-Process tab, click *Preview Pre-Processed Layout*.

For EMX models, the preview of the selected model is displayed in the EM Preview of Model <model-name> window.

It shows all the shapes, including the clustered vias and simplified shapes, that are saved in the `.clf` file. In addition, it shows the impact of cutting boundaries and scaling factors on the selected shapes.

If you have specified pre-processing settings, the preview also shows how the layout will be modified according to those rules and settings. You can make any necessary changes before submitting the model for simulation.

Related Topics

Specifying Shape Simplification Options for IC Layouts

Protecting Layout Objects from Shape Simplification

Pre-Process Tab

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

3

Running EM Extraction

This topic describes how to configure settings for running EM extraction for IC and package layouts using the EM solvers integrated with the Electromagnetic Solver assistant.

- Configuring Settings for Electromagnetic Simulations
- Distributing Simulation Runs to Multiple Resources
- Running a Simulation using Clarity 3D Solver
- Running the EMX Simulator From the Command Line
- Running an EMX Simulation (IC Layouts Only)

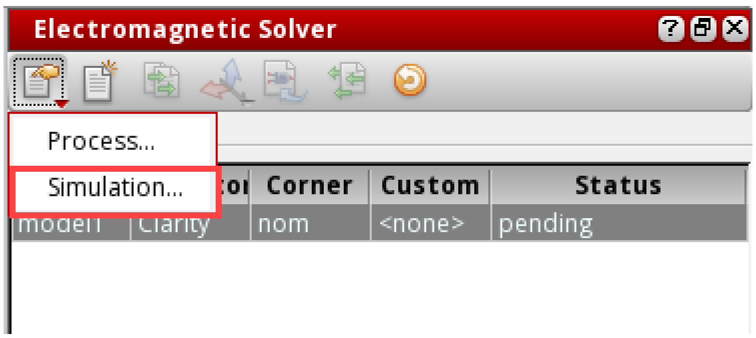
© 2023 Cadence Design Systems, Inc.

Configuring Settings for Electromagnetic Simulations

Before running an electromagnetic simulation, use the Simulation Settings form to configure the settings to be used for the simulator you want to use.

To configure the setting for Electromagnetic simulations:

1. Choose *Settings – Simulation* on the toolbar of the Electromagnetic Solver assistant to open the Simulation Settings form.

   The Simulation Settings form is displayed.

2. On the *General* tab, specify the general settings used by all simulators.
3. On the *Clarity* tab, specify the options to be used by the Clarity 3D Solver simulator.
4. On the *EMX* tab, specify the options to be used by the EMX simulator.
5. On the *LVS* tab, specify the options to be used by the LVS flow.
6. Click *OK* to close the form.

In a new session, the fields of this form show default values. You can modify the values to suit the requirements of your model. At any time, you can reset the fields to the default values by clicking *Default* on the form.

Related Topics

Simulation Settings Form

Overriding the Default Simulation Settings

Saving Simulation Settings

© 2023 Cadence Design Systems, Inc.

Overriding the Default Simulation Settings

By default, the Simulation Settings form shows the preset default value for each field. Depending on the model requirements, you can customize the default values that are loaded when you create a new model or reset the settings for an existing model. You can specify a different set of default simulation settings for an IC or package design.

To customize the default simulation settings:

1. Create a settings file with a list of settings for which you need to override the default values.

   To customize the settings for designs with different fabric types, create a separate file for each type.

2. For a design of IC fabric type, set the `vem.ic defaultSettingsFile` environment variable to specify the path to the file you created in step 1.

   Similarly, for a design with the package fabric type, set the `vem.package defaultSettingsFile` environment variable to specify the path to the file in which you specified the default values.

After you set the `vem.ic defaultSettingsFile` or the `vem.package defaultSettingsFile` environment variable, any cellview in which you open the Electromagnetic Solver assistant for the first time automatically uses the overridden values.

To override the default values for an existing model:

1. Choose *Settings – Simulation* from the toolbar of the Electromagnetic Solver assistant to open the Simulation Settings form.

2. Click *Default* on the form.

   All the fields in the form are reset to use the modified default settings specified in the file. Any existing default is also modified to use the new default value.

Related Topics

Contents of a Simulation Settings File

Simulation Settings form

© 2023 Cadence Design Systems, Inc.

Contents of a Simulation Settings File

You can create a custom text file to override the default simulation settings. This file contains a separate list for each tab of the Simulation Settings form. However, it is not mandatory to specify a list for each tab. You can specify a list for only that tab for which you want to override values. For example, if you need to use custom default values for the EMX simulator only, specify only one list for EMX.

All possible lists with the preset default values are given below.

```
("General"
    ("AdvancedFreqSweep" t)
    ("ExplicitDCSolution" t)
    ("FreqMin" 0)
    ("FreqMax" 10e9)
    ("SamplingType" "Linear")
    ("LogPointsPerDecade" 10)
    ("TransitionFreq" 10e6)
    ("LinearFreqStepSize" 10e6)
    ("MaxCPUs" 0)
)

("Clarity"
    ("SolutionFreq" 10e9)
    ("MaxAdapMeshIteration" 50)
    ("AdapRefinePercentage" 10)
    ("TargetDeltaS" 0.02)
    ("MinAdapIteration" 1)
    ("MinConvergedIteration" 1)
    ("MetalType" "Metal_Inside")
    ("BasisFuncOrder" "FIRST")
    ("dxPlusSize" 1000)
    ("dxPlusCondition" "Perfect electrical conductor")
    ("dxMinusSize" 1000)
    ("dxMinusCondition"  "Perfect electrical conductor")
    ("dyPlusSize" 1000)
    ("dyPlusCondition"  "Perfect electrical conductor")
    ("dyMinusSize" 1000)
    ("dyMinusCondition"  "Perfect electrical conductor")
    ("dzPlusSize" 1000)
    ("dzPlusCondition"  "Approximately open")
    ("dzMinusSize" 100)
    ("dzMinusCondition" "Approximately open")
    ("ConformalOuterBox" t)
    ("DielectricSize" 0)
    ("DielectricCondition" "Perfect electrical conductor")
    ("UseSignalNetMaxEdgeLength" t)
    ("SignalNetMaxEdgeLength" -1000)
)

)

("EMX"
    ("EdgeMesh"              1)
    ("Thickness"             1)
    ("ViaMerge"              0)
    ("ThreeDMetals"          "*")
```


© 2023 Cadence Design Systems, Inc.

```
    ("ViaCapacitance"           "")
    ("ViaInductance"            "")
    ("Radiation"                nil)
    ("EMXGDSViewCmdLineOptions" "")
    ("EMXCmdLineOptions"        "")
    ("GDSViewCmdLineOptions"    "")
   )
```

Example

To use a common set of EMX and GDSView command-line options for all EMX models, create a settings file as shown below.

```
("EMX"
    ("EMXCmdLineOptions"        "list-of-command-line-options")
    ("GDSViewCmdLineOptions"    "list-of-command-line-options")
   )
```

If you provide the path to this file by using the `vem.ic defaultSettingsFile` environment variable, cellviews will automatically use the command-line options given in this file when you open the Electromagnetic Solver assistant for the first time.

© 2023 Cadence Design Systems, Inc.

Saving Simulation Settings

Whenever you run an electromagnetic simulation for a model, the tool also saves the simulation settings used for that model in a text file in the run directory.

At any point, you can also save the settings from the Simulation Settings form to a file.

To save the settings:

1. In the Simulation Settings form, click *Save*.

   The Save Simulation Settings form is displayed.

2. Specify the file name.
3. Click *Save*.

The simulation settings are saved in the ASCII text format in the specified file.

You can edit the file and load it later in the same or a different cellview to reuse the saved simulation settings.

Related Topics

Contents of a Simulation Settings File

© 2023 Cadence Design Systems, Inc.

Loading Simulation Settings

To load the settings from a file into the Simulation Settings form:

1. In the Simulation Settings form, click *Load*.

   The Load Simulation Settings form is displayed.

2. Browse and select the file in which settings are available.
3. Click *Load*.

The imported simulation settings are displayed in the fields of the Simulation Settings form.

If the source file contains values for only a few fields, the values of other fields are retained. If the details loaded from the text file are not the settings for the form. an error is shown in CIW.

© 2023 Cadence Design Systems, Inc.

Layout Customization Options

When you run a simulation, the Electromagnetic Solver assistant saves the model information in a .clf file and sends it to the simulator. During this process, the tool might also run layout customization required for simulation.

You can set the following variables to influence the layout customization:

- `simplifyToSides`: Specifies the number of points to be used for the polygon shapes. While creating a .clf file, Virtuoso converts all circle and ellipse shapes found in the IC layout to polygons to avoid any potential loss of information that leads to missing shapes. The default shape of such polygons is hexagon.
- `discretizationMaxError`: Overrides the error limit automatically derived by the tool. For package layouts, Virtuoso converts curved shapes to polygons before sending the layout details to the EM solver. While doing this conversion, that is discretization, Virtuoso considers the maximum acceptable error limit to identify the number of polygons to be created. A high acceptable error limit, which also means a high error tolerance, results into fewer polygons as compared to a low error tolerance. By default, the tool takes the maximum acceptable error limit from the constraints set for the layout. It then prints the constraint name and the derived error limit in the CIW.

Related Topics

discretizationMaxError

simplifyToSides

© 2023 Cadence Design Systems, Inc.

Distributing Simulation Runs to Multiple Resources

Running simulations for large designs can be resource-intensive. Therefore, you can distribute the task to multiple computers in a computer farm by specifying a Distributed Resource Management System (DRMS) command. A DRMS command is the command submitted to distribute jobs to a workload job scheduling software, such as LSF or OpenLava.

To distribute the runs for electromagnetic extraction, do one of the following:

➡ Specify the `bsub` DRMS command by using the `dsmsCommand` environment variable.

➡ Specify the `bsub` DRMS command in the *Prefix Solver Commands with* field on the *General* tab of the Simulation Settings form

Virtuoso prepends the DRMS command to the command that launches the solver. After the run is successfully complete, the solver collates the results and writes those into a touchstone file.

Related Topics

clusterVias

© 2023 Cadence Design Systems, Inc.

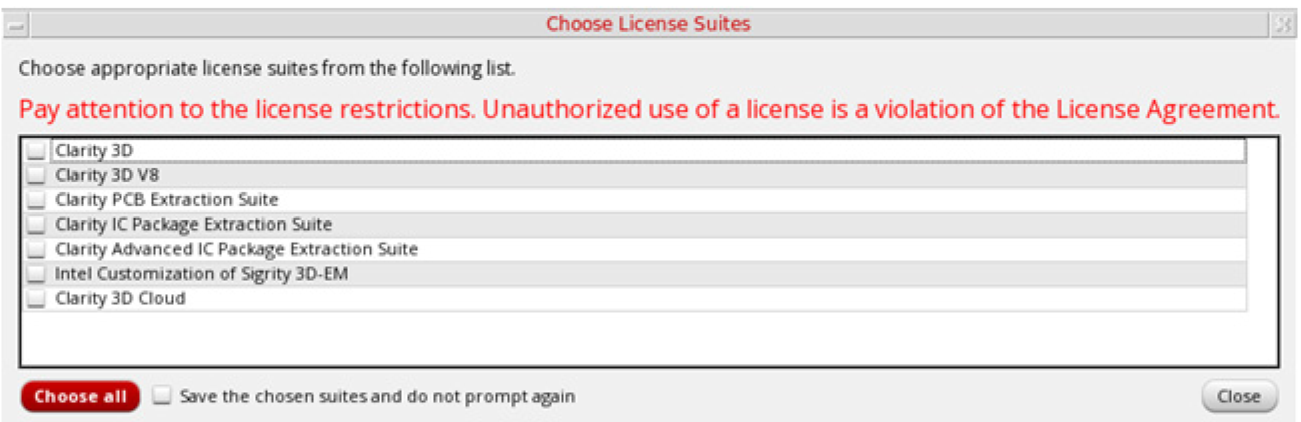
Running a Simulation using Clarity 3D Solver

Clarity 3D Solver uses a three-dimensional, full-wave, finite element method (FEM) field solver, and returns accurate simulation results.

To run a simulation using Clarity 3D Solver:

1. Review the simulation settings on the *Clarity* tab of the Simulation Settings form.
2. Select a model in the *Models* section.
3. Ensure that `Clarity` is selected in the *Simulator* column.
4. (Optional) Review the EM layout to verify the geometries being sent to the solver for simulation.
5. Choose *Simulator – Create and Edit* on the toolbar of this assistant.

   The Choose License Suites form is displayed.
6. Select *Clarity 3D* and close the form.



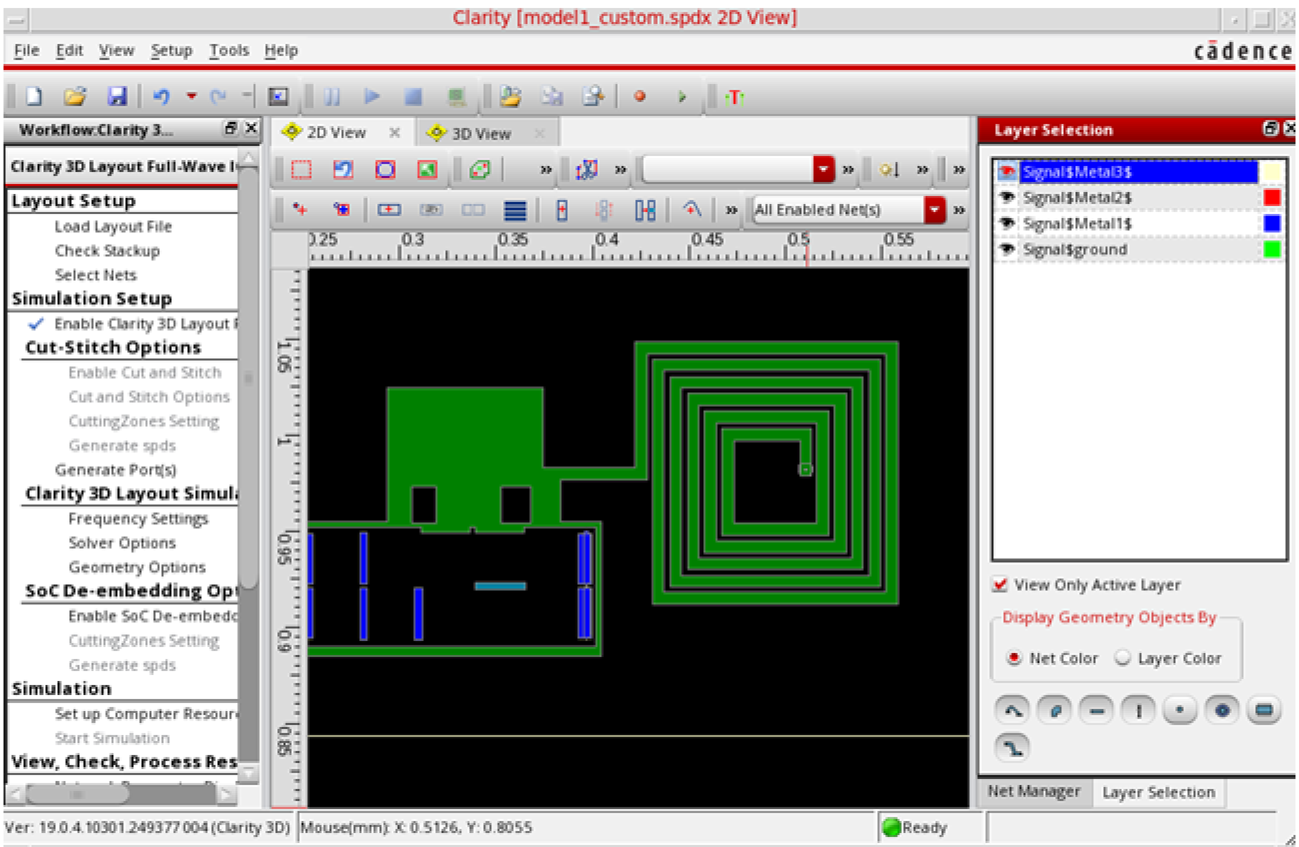
Tip

To use the same license suite every time Clarity 3D Solver is launched, use the interpolationMethod shell environment variable to specify a default license suite. When this variable is set, Virtuoso looks for the specified license before launching Clarity 3D Solver. If found, it does not display the Choose License Suites form.

After checking out the license, the tool does the following:

© 2023 Cadence Design Systems, Inc.

- ❑ Loads the model definition and creates the layout database that is ready for simulation.
- ❑ Saves the database in `.spd` files.
- ❑ Opens the Clarity window and loads the `.spd` file.
- ❑ Displays the instances and nets you selected for inclusion in the model in the Clarity window.

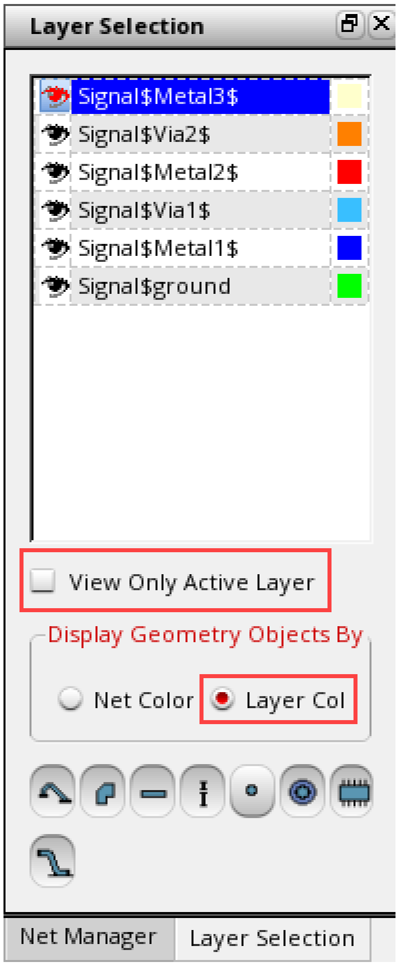


Virtuoso sends all the details of the shapes and nets included in the model to Clarity 3D Solver. When a hierarchical net has a different name at each level, it sends the name of only the top-level net.

7. Make the following changes in the Layer Selection assistant:
   - ❑ To view all layers, clear the *View Only Active Layer* check box.

© 2023 Cadence Design Systems, Inc.

- ❑ To identify the objects by layer color, select the *Layer Color* option in the *Display Geometry Objects By* group.

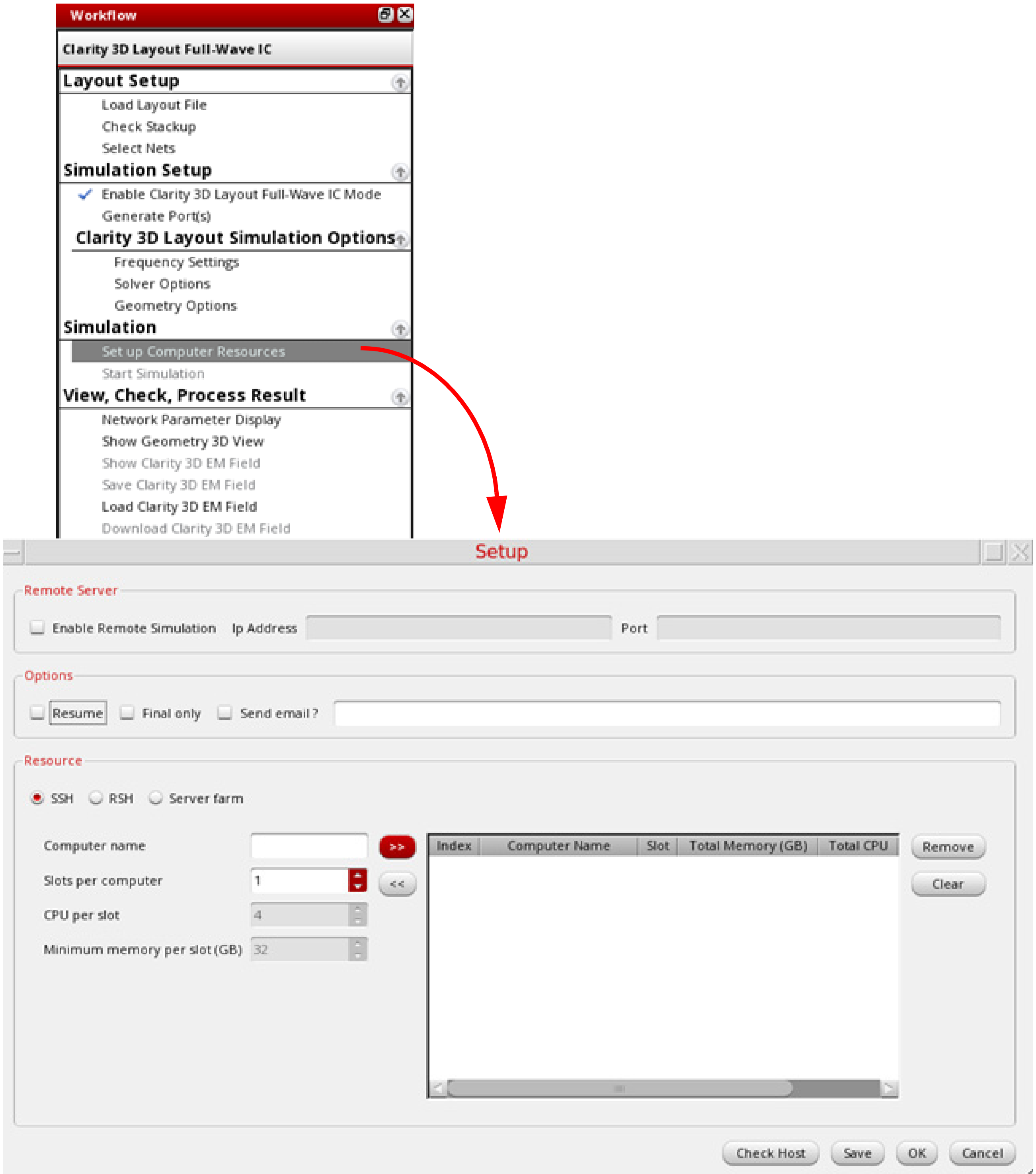


- ❑ By default, the top layer is visible, but you can select any other layer name to view the shapes on that layer.

8. Click *Set up Computer Resources* in the Workflow assistant to configure the setup.

© 2023 Cadence Design Systems, Inc.

The Setup form is displayed.



© 2023 Cadence Design Systems, Inc.

9. Depending on the resources you want to use for the simulation run, you can configure the settings to use a remote server farm or the localhost. However, ensure that the resource must have a minimum of 32GB memory per slot.

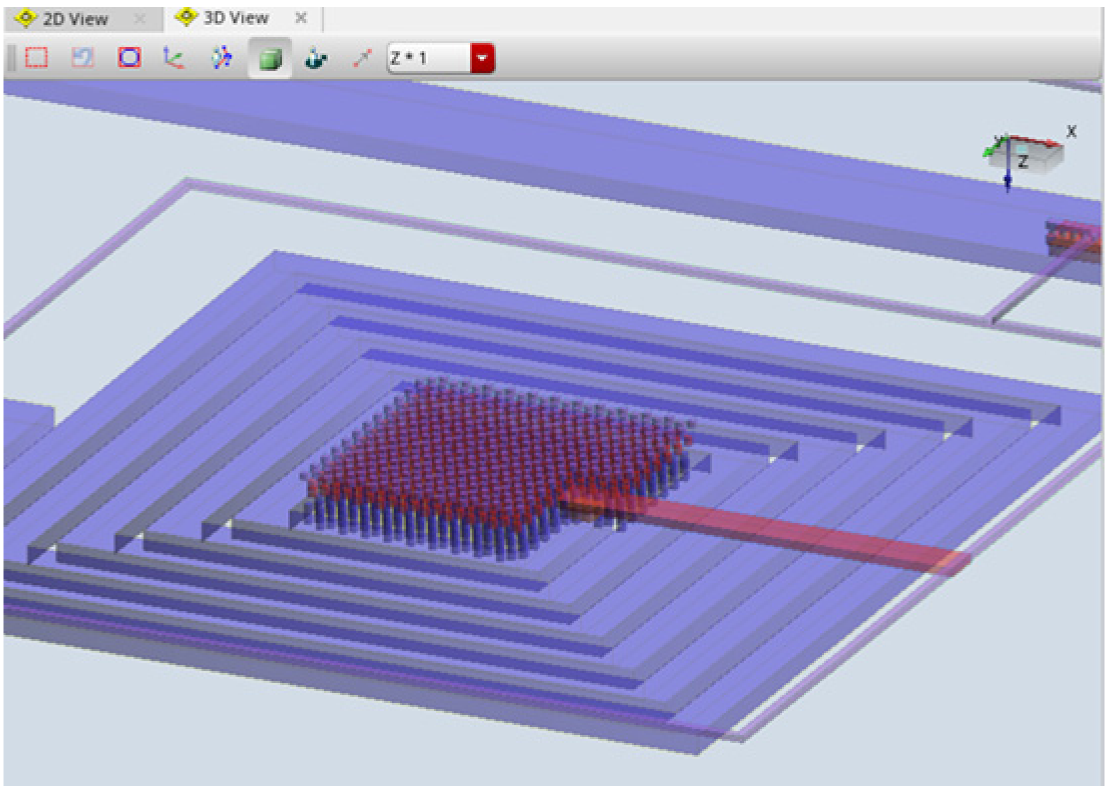
   For details about the settings on the Setup form, refer to *Chapter 3, Simulation Flow* in the *Clarity 3D Layout User Guide*.

10. After specifying the resources, click *Save* on the Setup form to save the setup.

    Once done, the setup is saved in the `~/.cadence/Clarity/configSave.json` file. This setup is automatically read from this file before running simulations.

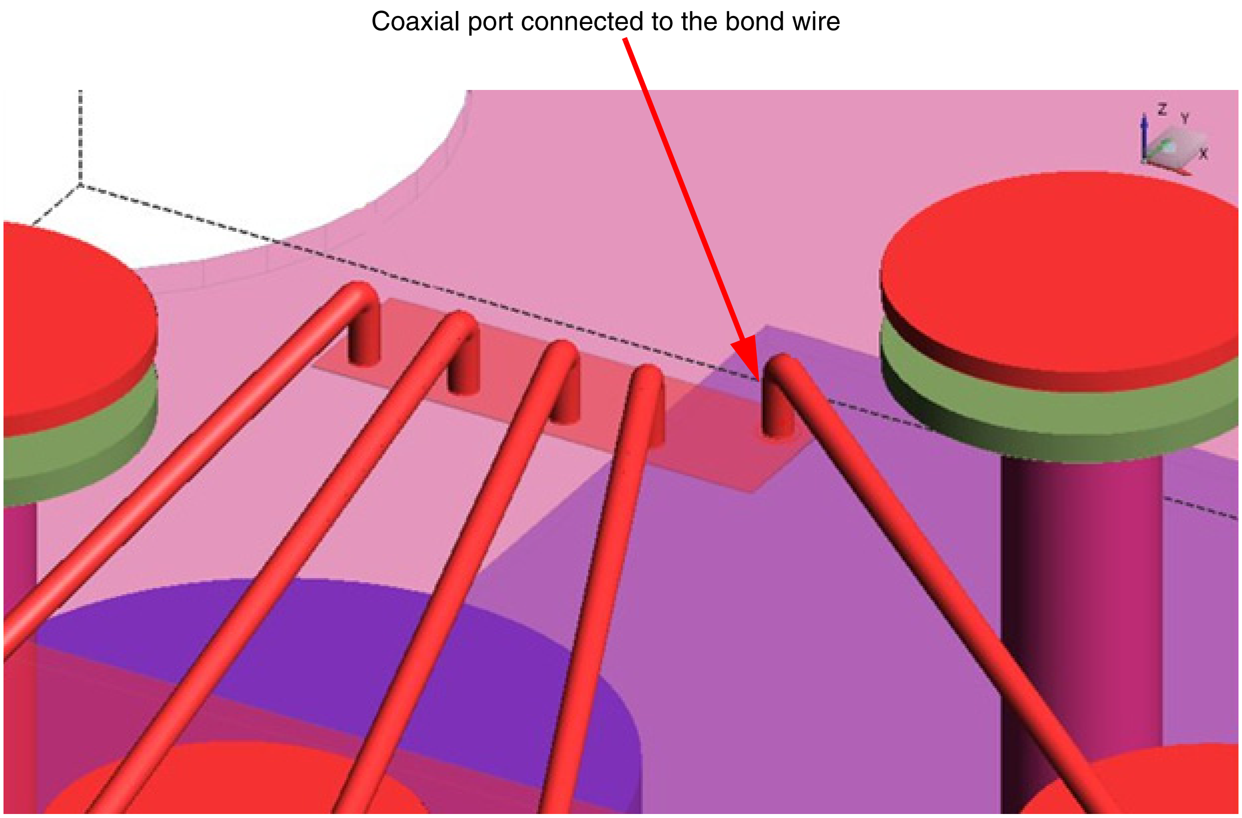
11. To review the model in 3D view, click *Show Geometry 3D View* in the *View, Check, and Process Result* section of the Workflow assistant.

    The following figure shows an example 3D view of an inductor selected for a model.



© 2023 Cadence Design Systems, Inc.

For bond wires included in models, Clarity creates a coaxial port for the net connected to the bond wire, as shown in the example given below.



12. Click *Start Simulation* on the toolbar of Sigrity Suite window to run simulation.

When the simulation is complete, the results are displayed in the *Network Display* tab.

13. To review the mesh created by Clarity, click *Show Clarity 3D EM Field* in the *View, Check, and Process Result* section of the Workflow assistant.

14. You can review the results in the *Network Display* tab of this window or in Virtuoso Layout MXL, and then continue with backannotation of results.

Related Topics

Creating Extracted Views from Models

© 2023 Cadence Design Systems, Inc.

Running an EMX Simulation (IC Layouts Only)

EMX Planar 3D Solver simulates high-frequency, RF, and mixed-signal integrated circuits. It lets to simulate large RF circuit blocks accurately and efficiently, characterize the behavior of passive components, and analyze the parasitics due to interconnects.

Important

To simulate a model using EMX and view the resulting mesh in Virtuoso 3D Viewer, you must ensure that the prerequisites are met.

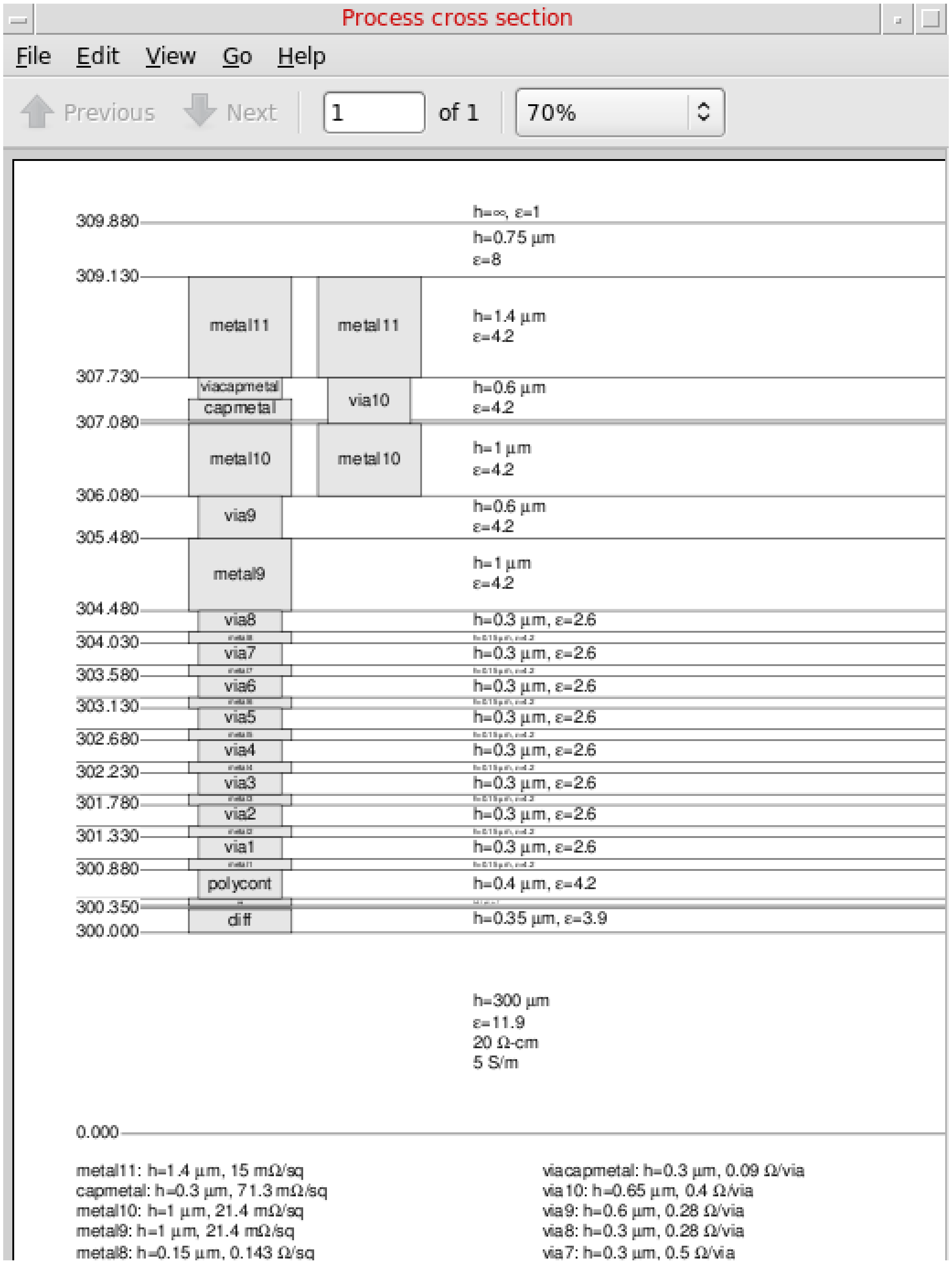
To run a simulation using the EMX solver, perform the following steps:

1. Set the emxProcessCornerDirectory environment variable to specify the path to the directory where `.proc` files with process corner settings are saved.
2. (Optional) Set the emxProcessDefaultCorner environment variable to specify the default process corner name.
3. (Optional) Set the emxLayerMap environment variable to specify the path to the layer map file to be used while sending models to the solver.
4. Review the simulation settings on the *EMX* tab of the Simulation Settings form.
5. In the Electromagnetic Solver assistant, create a new model in the *Models* section.
6. Select *EMX* in the *Simulator* column for the model. You can select multiple EMX models and simulate them in parallel.
7. Review the layer stackup defined in the `.emproc` files by choosing one of the following commands from the toolbar of the assistant:
   - *View EMX Scaled Stackup*
   - *View EMX Unscaled Stackup*

   The tool opens the PDF viewer and displays the scaled or unscaled layer stackup depending on the chosen command.

© 2023 Cadence Design Systems, Inc.

An example layer stackup is shown below.



8. On the *Selection* tab of the Electromagnetic Solver assistant, select objects to be included in the model.

   To include the complete layout in the model, select the *Extract Full Cellview* check box.

© 2023 Cadence Design Systems, Inc.

9. On the *Ports* tab of the Electromagnetic Solver assistant, add port details. To ensure correctness, you can also validate the ports by using the *Validate Ports* command on this tab.

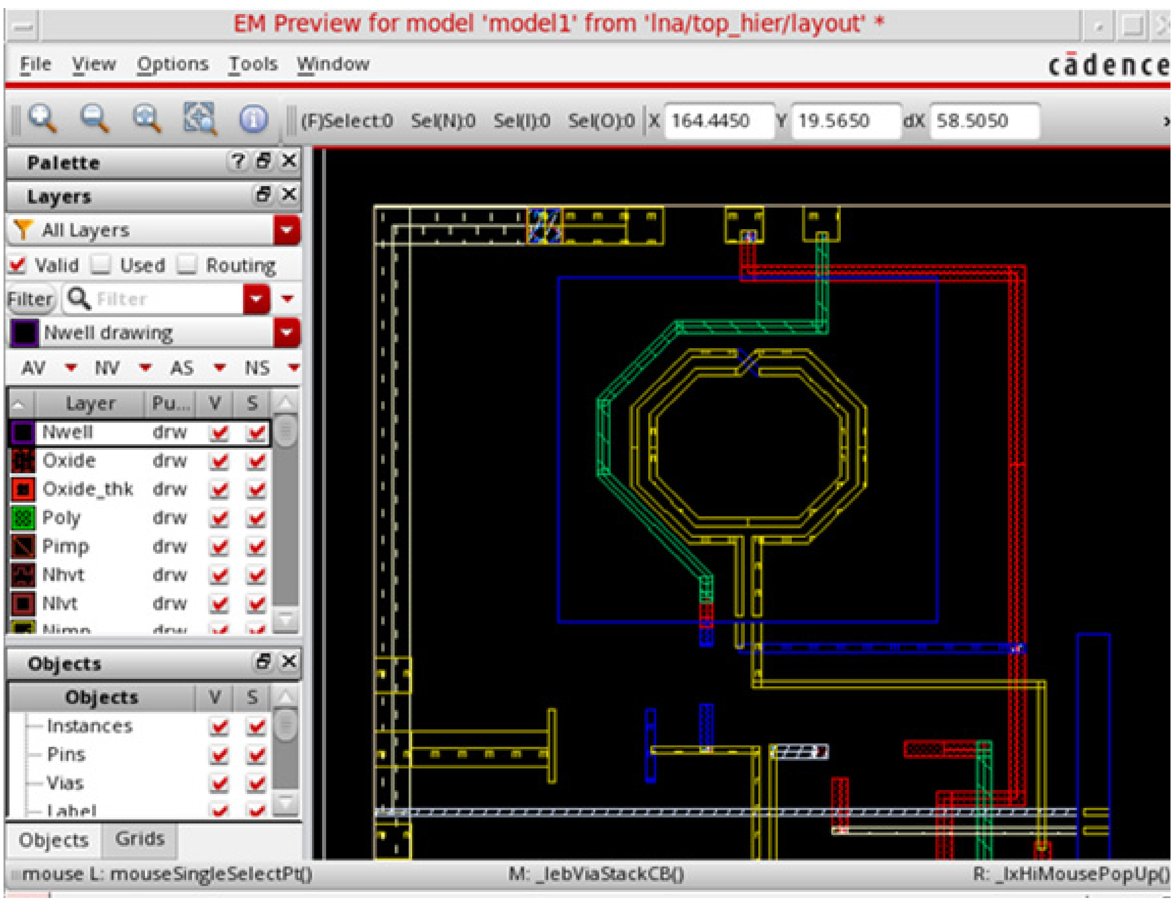
   Only unreferenced ports are created for EMX models.

10. (Optional) Specify pre-processing options.

11. (Optional) Do one of the following to preview the model and verify the objects and shapes being sent to the solver for simulation:

   ❑ Click *Preview EM Layout* on the *Pre-Process* tab.

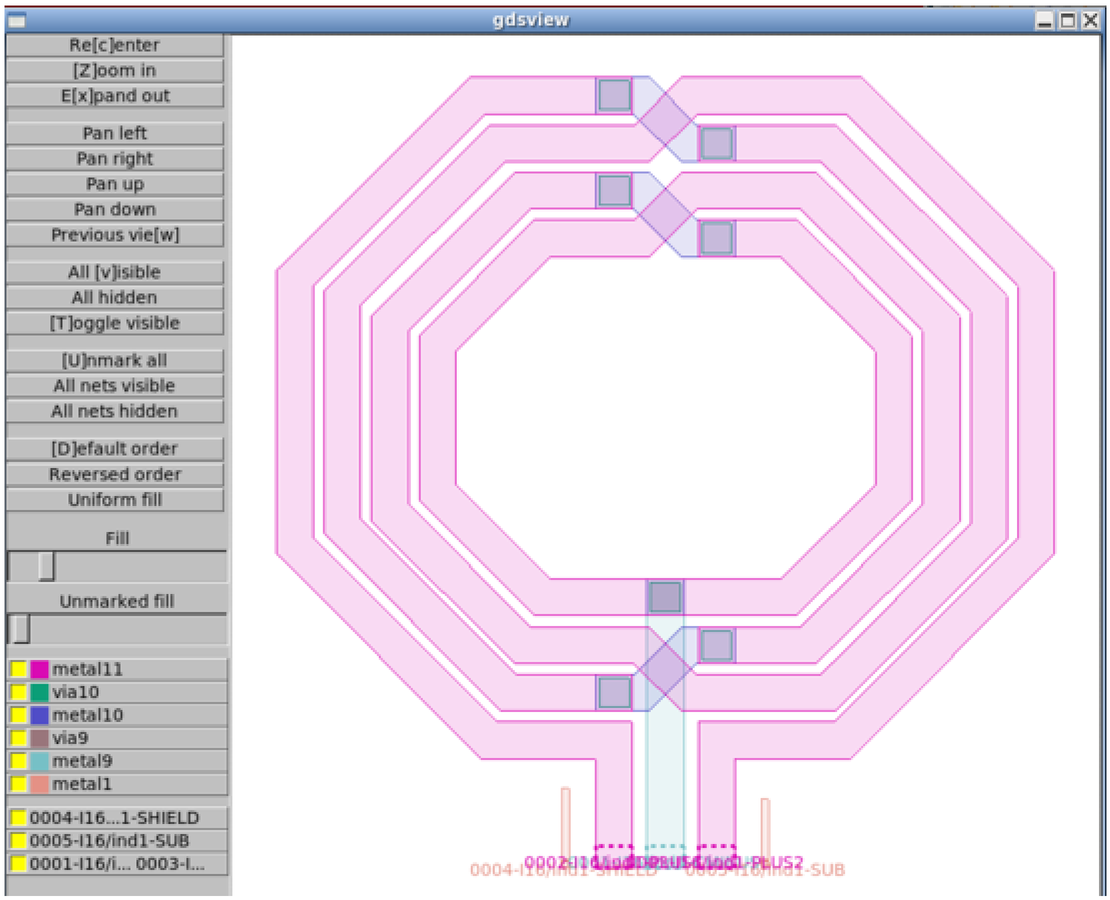
   The tool displays the preview of the selected shapes in the EM Preview of Model <model-name> window. An example is shown below.



   ❑ Choose *Open Simulator – Preview Geometry* on the toolbar of the Electromagnetic Solver assistant.

© 2023 Cadence Design Systems, Inc.

EMX uses its own viewer, GDSview, to display the shapes selected for the model. An example is shown below.



12. (Optional) Choose *Open Simulator – Generate Mesh* on the toolbar to generate a mesh for the model.

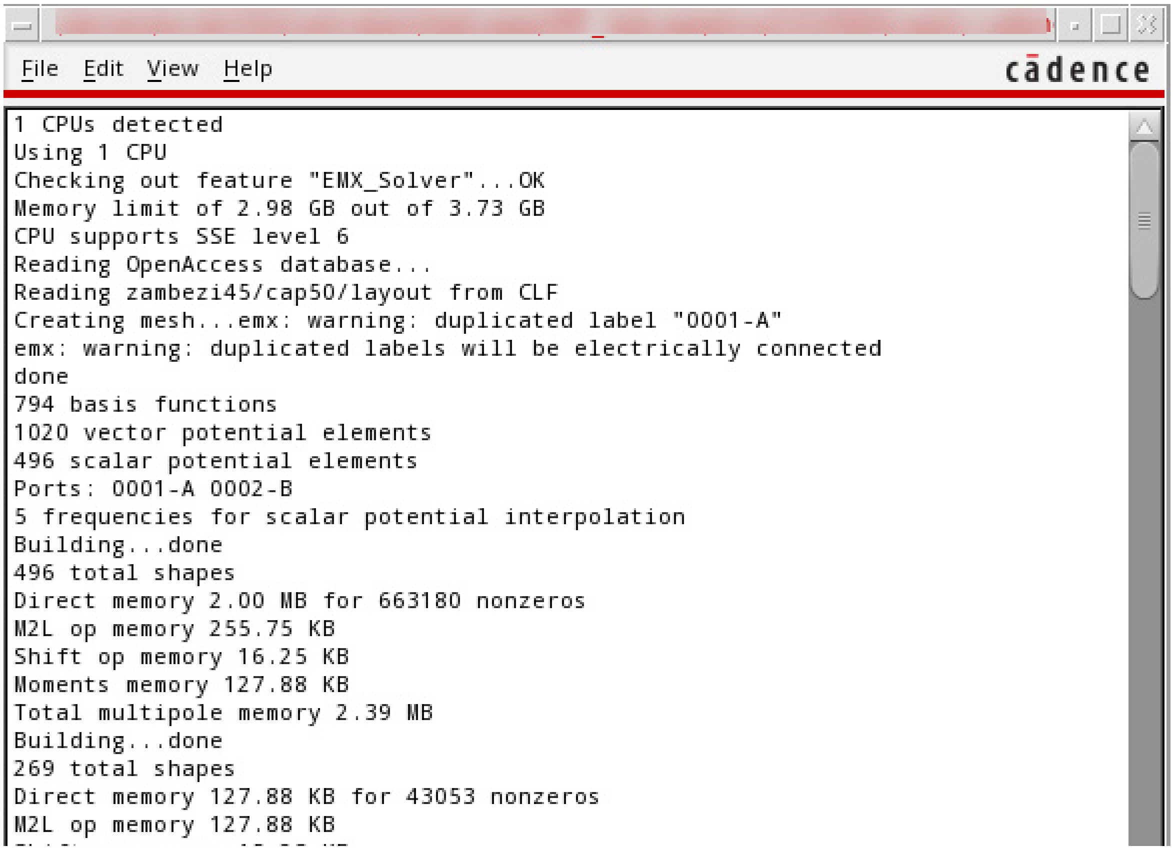
The tool sends the objects to the EMX solver, which creates a 3D model for the selected objects by identifying smaller sections that together form a mesh. If all the ports are valid and a mesh is successfully created, EMX saves the mesh in a `.vmesh` file for your layout. Virtuoso then launches Virtuoso 3D Viewer to display the mesh.

13. Choose *Open Simulator – Mesh and Simulate* on the toolbar to run simulation for the model.

EMX recreates the mesh and runs simulation. During the simulation run, EMX uses the settings specified on the *General* and *EMX* tabs of the Simulation Settings form to compute the electric field in each section of the mesh.

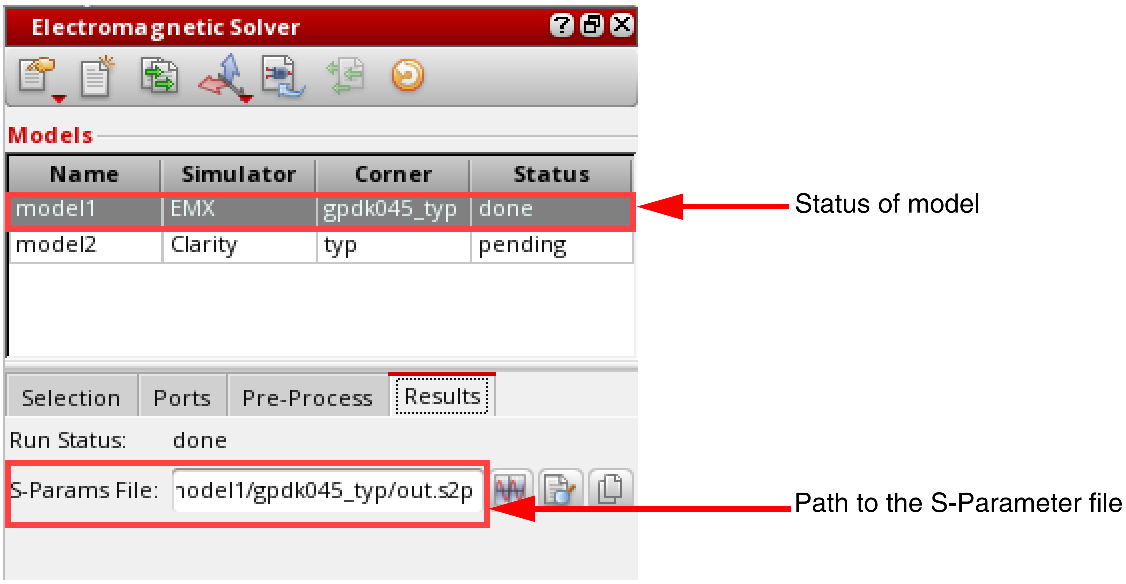
© 2023 Cadence Design Systems, Inc.

A log window is displayed to show the run status and summary of progress.



© 2023 Cadence Design Systems, Inc.

14. After the run is complete, the status of the model changes to `done` and the results are loaded into the *Results* tab of the assistant. The path to the s-parameter file created by EMX is displayed in the *S-Params* field on this tab.



15. Click *Create S-Parameter View* on the toolbar of this assistant to create an extracted view for the model.

Related Topics

Pre-processing options

Reviewing the Mesh Created by EMX in Virtuoso 3D Viewer

Virtuoso 3D Viewer

Creating Extracted Views from Models

© 2023 Cadence Design Systems, Inc.

Running the EMX Simulator From the Command Line

You can run the EMX simulator directly from the command line. For example, if you need to copy the CLF file to a different host with more resources, you can use the following CLF-specific executables from the command line.

```
emx_clf file.clf [list of emx command line options]
gdsview_clf file.clf [list of gdsview command line options]
```

The first argument for the executables is a `.clf` file. The other arguments will be passed to the actual binary.

To use the executables, set EMX and GDSview in the `$PATH`. The executables use the path to a `.clf` file and invoke EMX or GDSview on it. `emx_clf` invokes the EMX solver and `gdsview_clf` invokes the EMX 2D layout viewer.

Related Topics

Running an EMX Simulation (IC Layouts Only)

Running a Simulation using Clarity 3D Solver

© 2023 Cadence Design Systems, Inc.

Reviewing the Mesh Created by EMX in Virtuoso 3D Viewer

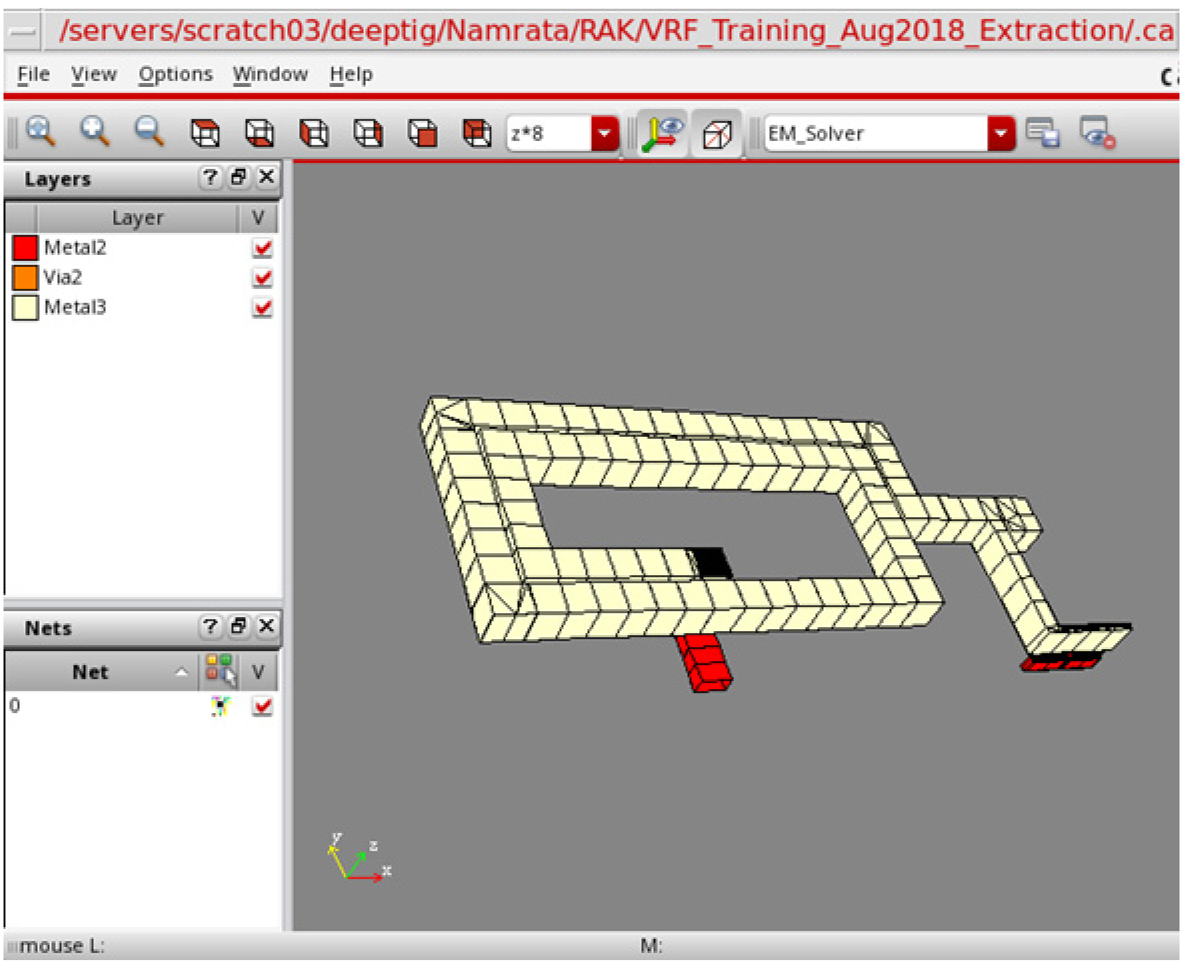
Before running a simulation for a model, it is recommended that you generate and review the mesh for the datasets selected for the model.

To review the mesh for a new model:

1. Choose *Open Simulator – Generate Mesh* on the toolbar of the Electromagnetic Solver assistant.

   This command creates a mesh and saves it in a `.vmesh` file and then it launches Virtuoso 3D Viewer to display the generated mesh.

   Alternatively, if the mesh already exists, use the *Open Simulator – View 3D Mesh* command on the toolbar to open the most recent `.vmesh` file in the 3D Viewer.



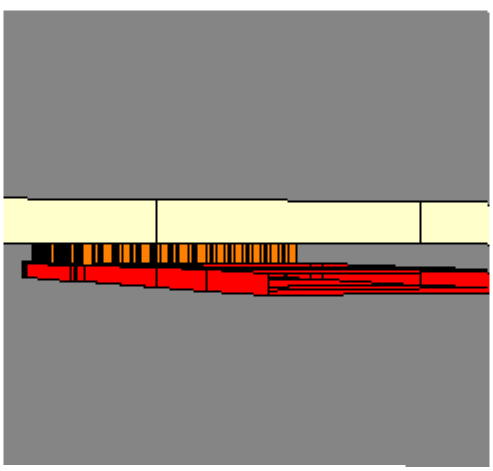
© 2023 Cadence Design Systems, Inc.

2. By default, the 3D Viewer displays the mesh in the canvas as a flat view. To view the mesh in 3D, click and hold anywhere in the canvas and drag the pointer slightly in the direction of interest.

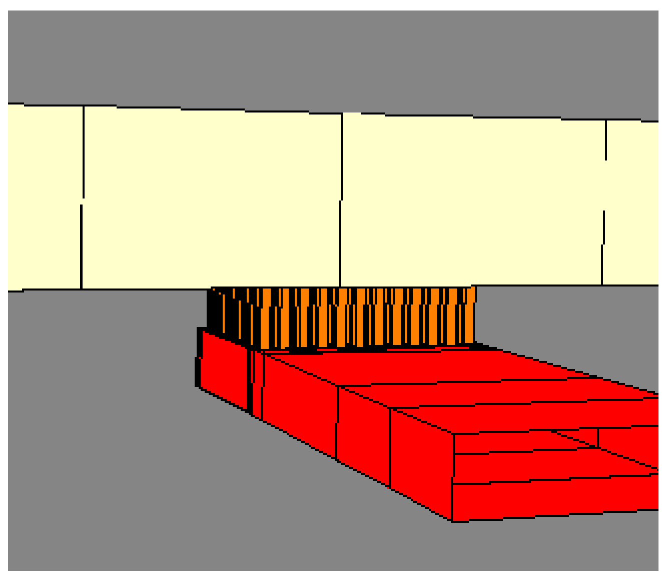
3. Look for any unintentional opens or shorts in your structure. You can also check how well you have configured your circuit for simulation. The most common problem is over-meshing a structure, which can result in a significantly longer simulation run time.

4. Use the following commands or controls available in the 3D Viewer:

   - Use the View toolbar to change the zoom settings or to view the mesh from different directions, such as, top view or right view. By hiding, showing, rotating, and zooming, you can focus on specific items you want to observe in the mesh.
   - Rotate the orientation axes at the bottom left of the canvas to set the direction from where you want to look at the mesh.
   - To to see thin layers clearly or to view the mesh between the layers, increase the z-axis by choosing a scaling factor on the View toolbar.
   - The following examples shows a comparison of layers in the default view and with the z-axis with 4x scaling.



3D View with default z-axis setting

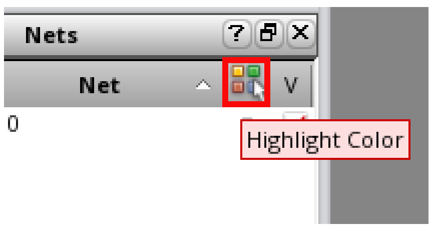


3D View with 4x z-axis scaling

   - Use the Display Options toolbar to control the visibility of wire frames or orientation axes on the canvas.

© 2023 Cadence Design Systems, Inc.

- Use the check boxes in the Layers assistant to display or hide layers and the shapes on those layers.
- Use the check boxes in the Nets assistant to control the visibility of nets and the metal islands connected to them.
- Use the *Highlight Color* command on the Nets assistant to change the coloring mode for nets. By default, the nets and the metal islands connected to them electrically are displayed in the layer color. You can toggle this command to use a unique color for each net and the metal island connected to it.



Related Topics

Virtuoso 3D Viewer

View toolbar

Display Options toolbar

Nets assistant

Layers assistant

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

4

S-Parameter Model Extraction

After the simulation is complete and you refresh the Electromagnetic Solver assistant, the status of the model is changed to `Done` and the S-Parameters file is returned to Layout MXL.

Before extracting the S-Parameters into an extracted view, you can view and analyze the results. If the models are ready to be used in the schematic view for further analysis, save them in an extracted view, which contains n-ports with S-Parameters. When you place these models in the schematic view, it deletes the existing instances and models, and creates instances or nets using the components from the extracted view. It further re-establishes the connections of the extracted nets and instances using n-ports.

Depending on the type of layout or the type of devices you selected for a model, the Electromagnetic Solver assistant lets you create S-Parameter models in different ways.

- To extract everything, minus very few excluded cells, use full cellview extraction.

If you want to extract only the selected nets, you can choose from the following options:

- If the selected nets are XL compliant, you can use the XL mode.
- If the selected nets are not XL compliant or you want to stitch into a Smart View, use the LVS flow.

Related Topics

Validating S-Parameter Data Saved by an EM Simulation

Creating Extracted Views from Models

Extracting S-Parameter Models from Full Cellviews (IC Layout Only)

Creating an S-Parameter Model for the Complete Layout (Passive Devices)

Extracting Nets with EMX Using the LVS Flow

Creating S-Parameter Cellviews for Models

© 2023 Cadence Design Systems, Inc.

Creating an S-Parameter Model for a Layout-Driven Flow

Extracting Models for a Cross-Fabric Design

© 2023 Cadence Design Systems, Inc.

Validating S-Parameter Data Saved by an EM Simulation

After the simulation is complete and you refresh the Electromagnetic Solver assistant by Clicking *Refresh* on the toolbar, the status of the model is changed to `Done`. The tool displays the path to the `.s2p` or `.snp` S-parameter file on the *Results* tab of this assistant.

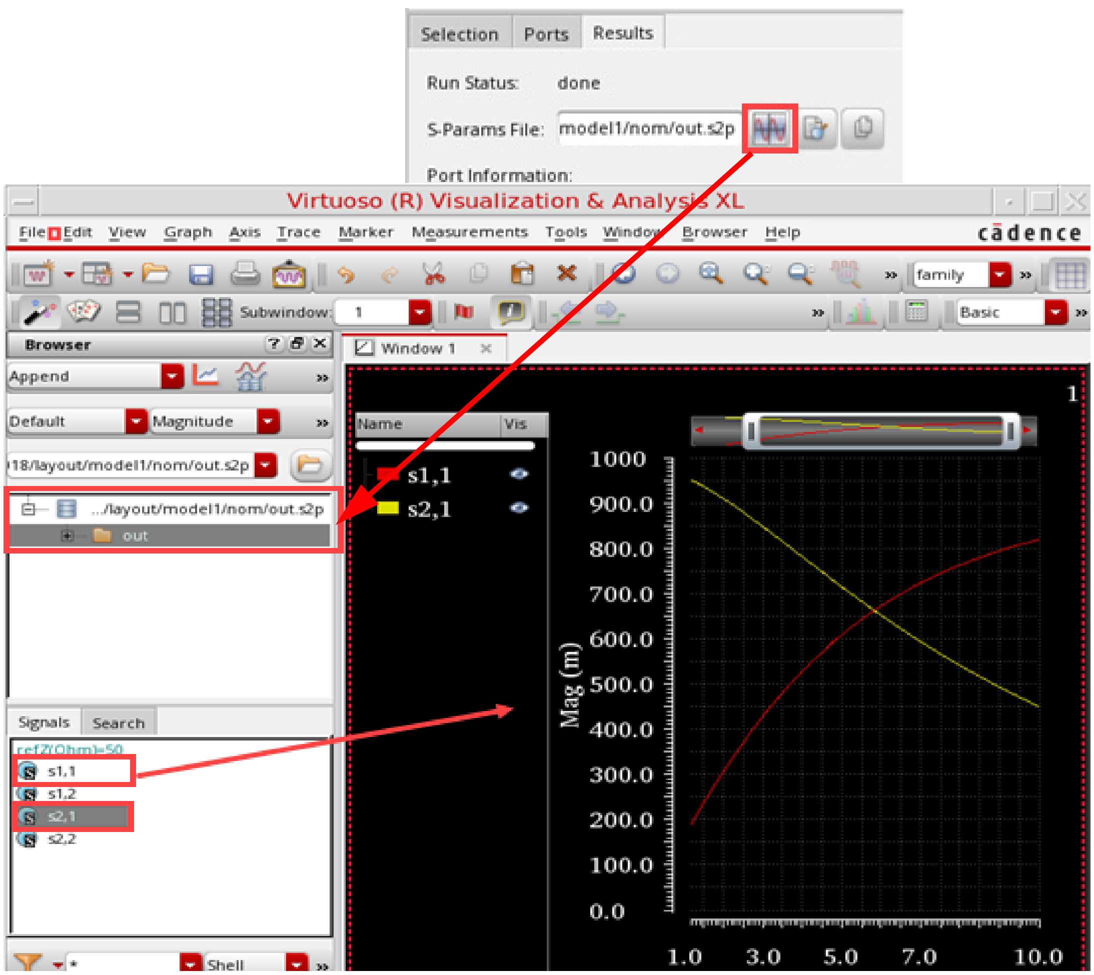
For validation, you can open the results of an EM simulation in multiple ways described below.

Plotting S-Parameter Data in Virtuoso Visualization and Analysis XL

To plot and analyze S-parameter data in Virtuoso® Visualization and Analysis XL:

1. Click *Open in Results Browser* to open the S-parameter file in Results Browser.
2. In the Results Browser, select specific signals.

   The signals are plotted in the graph window.



© 2023 Cadence Design Systems, Inc.

Viewing S-Parameter Data in a Text Editor

To review all data points in the text format, you can open the S-parameter file in a text editor.

➡ Click *Open in text viewer* on the *Results* tab.

The results are opened in the default text editor set for Virtuoso.

Viewing S-Parameter Data in any Other Application

As the simulation results are a collection of data points, you can review the data in any other third-party application.

➡ Click *Copy path to clipboard* on the *Results* tab.

The results are copied to the clipboard from where you can copy those to another application.

© 2023 Cadence Design Systems, Inc.

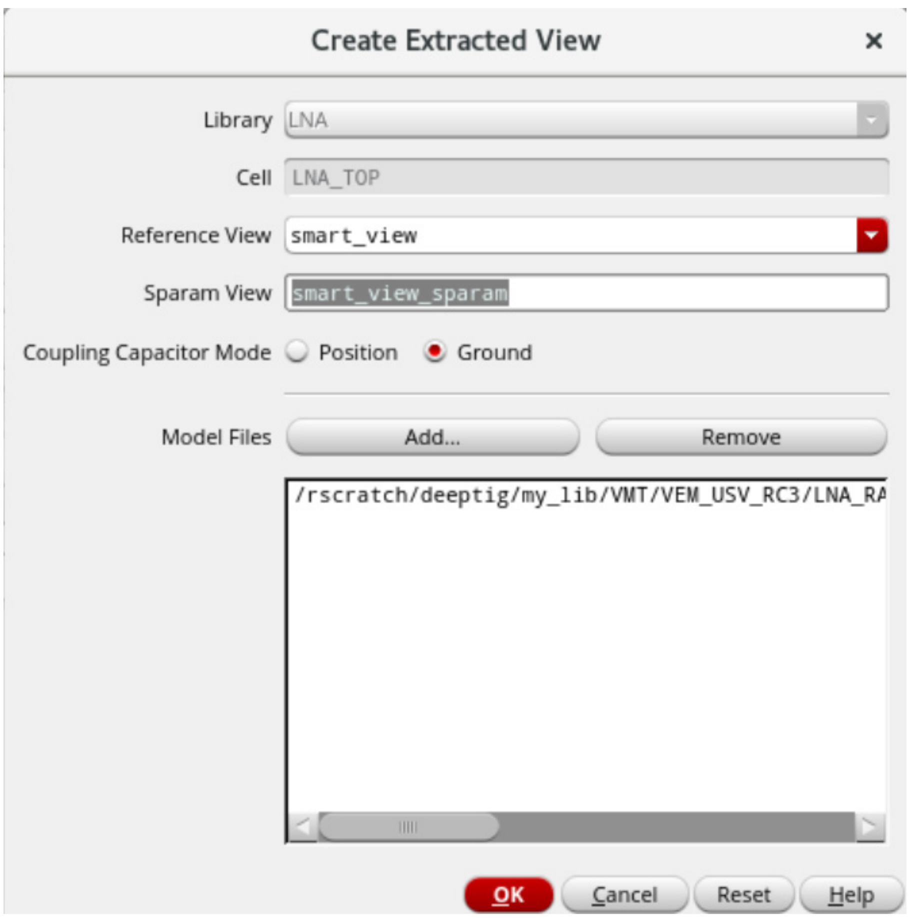
Creating Extracted Views from Models

After validating the EM simulation results, you can create of an extracted view. While creating an extracted view, Virtuoso creates a copy of the schematic and replaces the modeled devices with an n-port that points to the S-Parameters saved in the EM simulation results. It also saves the S-parameter file inside the target cellview.

To create an extracted view for a model selected in the Electromagnetic Solver assistant:

1. Select the models that meet the following conditions in the assistant.
   - ❑ The models must have status `Done`. If multiple models are selected, none of them can be full cellview models. If a full cellview model is selected, it must be the only model selected.
2. Click *Create Extracted View* on the toolbar of the Electromagnetic Solver assistant.

   The Create Extracted View form is displayed.



© 2023 Cadence Design Systems, Inc.

This form shows the library, cell, and view name for the extracted view.

3. In the *Reference View* drop-down list, choose the name of the view that you want to use as a connectivity reference to stitch the devices and nets from the S-parameter model. You can use a schematic view, Quantus Smart View, or another layout view as a reference.
4. (Optional) In the *Sparam View* field, specify a name for the extracted view to be created.

   The default name used for an extracted view is `emsolver_extracted`.
5. Specify *Position* or *Ground* to stitch a coupling capacitor when one of the two nets attached to the capacitor is included in the model.
6. Use the *Add* and *Remove* commands to modify the Model Files list so that it contains all the files to be included in the extracted view.
7. Click *OK*.

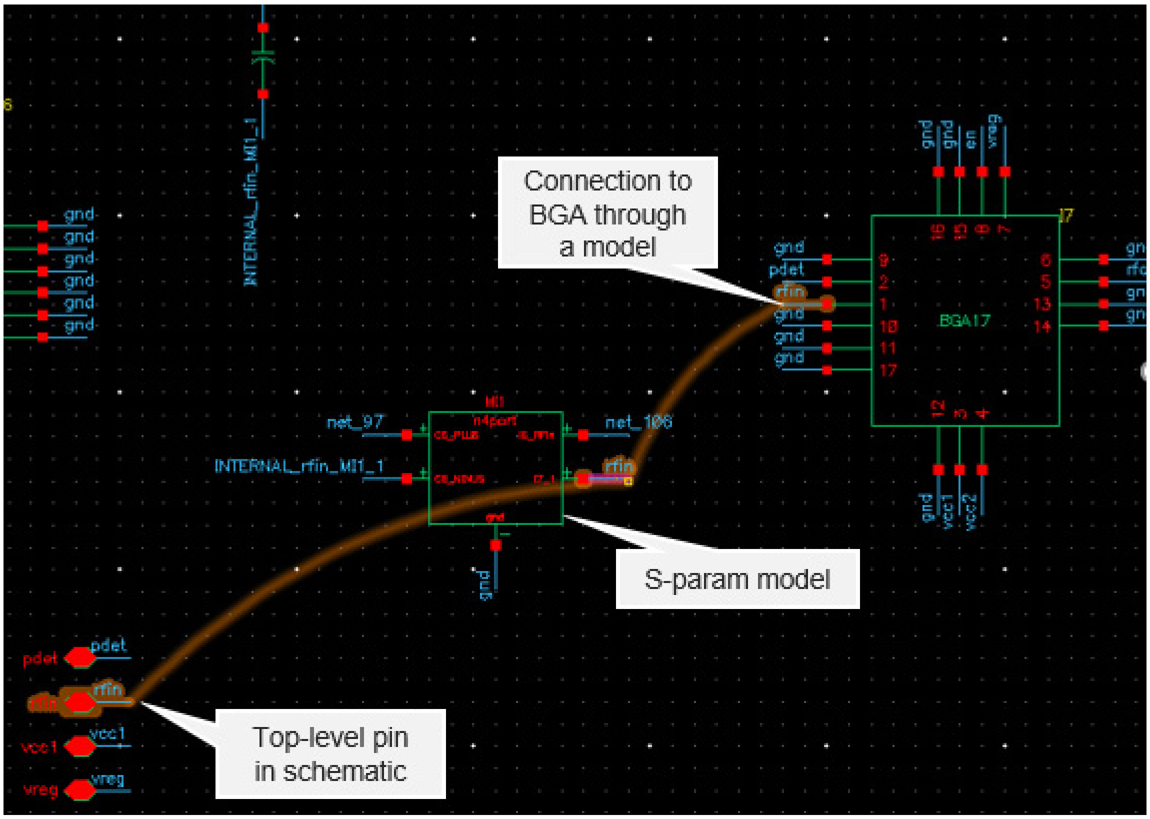
In non-interactive mode, the tool automatically places the device models in the schematic and saves the specified extracted view.

While placing the devices, the tool does the following:

- While stitching an S-param model into a package schematic view, the tool looks for package components and connects the external top-level pin in the schematic to the package component through the S-parameter. Connections from other device instances to the package instance are also recreated through the S-parameter.

© 2023 Cadence Design Systems, Inc.

In the following example, the top-level pin `rfin` is connected to the package instance `BGA17` through the S-param model.



- Checks and reports the following scenarios as errors:
  - Duplicate nets found in a single model
  - The model includes a net that is not found in the design
  - A single net is extracted into two different models and user passes both models to the extracted view engine.
- Does not report errors for any extra pins found in the schematic for which the property to ignore pins is set and no corresponding S-parameter port exists in the model. Common examples of such pins are the bulk or substrate pins of devices. If you set a property to ignore the extra pins, the tool does not report errors and retains the pins connected to the nets in the schematic.

  Note: You can use the `propsUsedToIgnoreObjs` environment variable to specify the name of the property to be used to ignore devices. Use the following command to get the property names:

```
envGetVal("layoutXL" "propsUsedToIgnoreObjs")
```


© 2023 Cadence Design Systems, Inc.

- If the reference view is a Smart View, the tool checks the value of the couplingCapMode environment variable to decide how to stitch a coupled capacitor when one of the two nets attached to the capacitor is included in the model. After stitching, the tool reports the total coupling capacitance value of each net that was in the model before and after the insertion of an n-port. It also reports the difference between the two values.

```
\o Total Coupling Cap For Net: net3 [pre]: "3.48426e-15"
\o    Total Coupling Cap For Sparam Net: INTERNAL_net3_MI1_1 [post]: "4.45292e-16"
\o    Total Coupling Cap For Sparam Net: INTERNAL_net3_MI1_3 [post]: "8.34068e-16"
\o    Total Coupling Cap For Sparam Net: INTERNAL_net3_MI1_7 [post]: "1.37775e-16"
\o    Total Coupling Cap For Sparam Net: net3 [post]: "3.97132e-16"
\o    Total Coupling Cap For Sparam Net: INTERNAL_net3_MI1_2 [post]: "2.68953e-16"
\o    Total Coupling Cap For Sparam Net: INTERNAL_net3_MI1_4 [post]: "5.48718e-16"
\o    Total Coupling Cap For Sparam Net: INTERNAL_net3_MI1_5 [post]: "4.02961e-16"
\o    Total Coupling Cap For Sparam Net: INTERNAL_net3_MI1_6 [post]: "4.4936e-16"
\o    Difference [ Pre - Post ] = "0"
```

- Checks for cellviews specified in the setNotEmbedded environment variable. The tool considers the instances of the specified cellviews as non-embedded components and excludes those from the extracted cellviews.

You can refer to the saved extracted view in ADE Explorer or ADE Assembler, and send it to Spectre for the re-simulation flow. After the simulation is run, compare the results saved for the schematic view with the results saved for the extracted view.

Note: To understand how to simulate S-Parameters in the time domain simulation, read the 7 Habits of Highly Successful S-Parameters application note on Cadence Learning and Support.

Related Topics

Resimulation flow

Create Extracted View Form

© 2023 Cadence Design Systems, Inc.

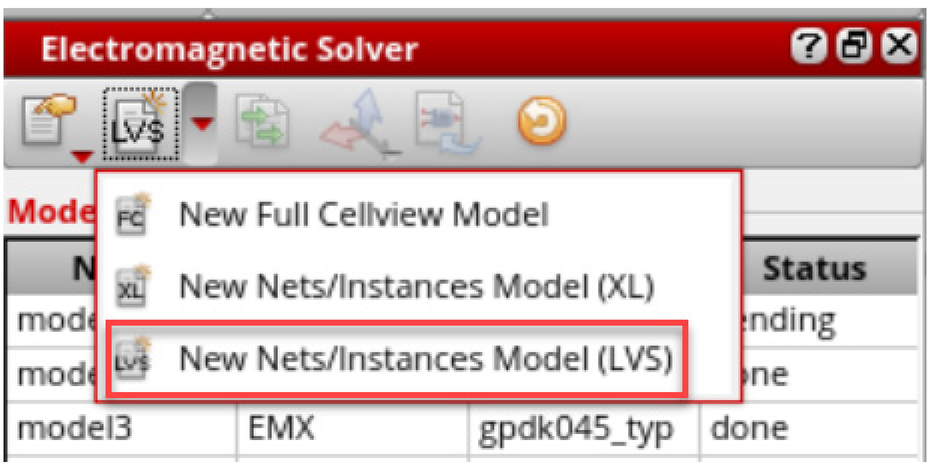
Extracting Nets with EMX Using the LVS Flow

This flow uses only the sign-off LVS, which ensures full synchronization with the Smart Views generated by Pegasus™ Verification System and Quantus™ Extraction Solution. However, the XL flow uses the Virtuoso XL extractor to extract the nets, which requires a properly configured XL and an XL-compliant layout. The LVS flow does not use XL extractor and does not require XL compliance. It only needs the output from a clean LVS run.

To perform LVS-based EMX extraction and Smart View stitching using nets:

1. Choose *New Nets/Instances Model (LVS)* from the drop-down list to create a new LVS-based model.

   A new LVS model is created. Such models have an *SVDB Directory* field at the bottom. The SVDB directory is created by the LVS tool.

2. Browse through *SVDB Directory*. Descend into the LVS run folder and select `svdb`.

   The SVDB database is loaded into memory. You can customize the *SVDB Directory* label as well.

   Note: For large designs, reading SVDB can take several minutes. However, it is a one-time step. Once the SVDB is in memory, it does not need to be read again until you rerun

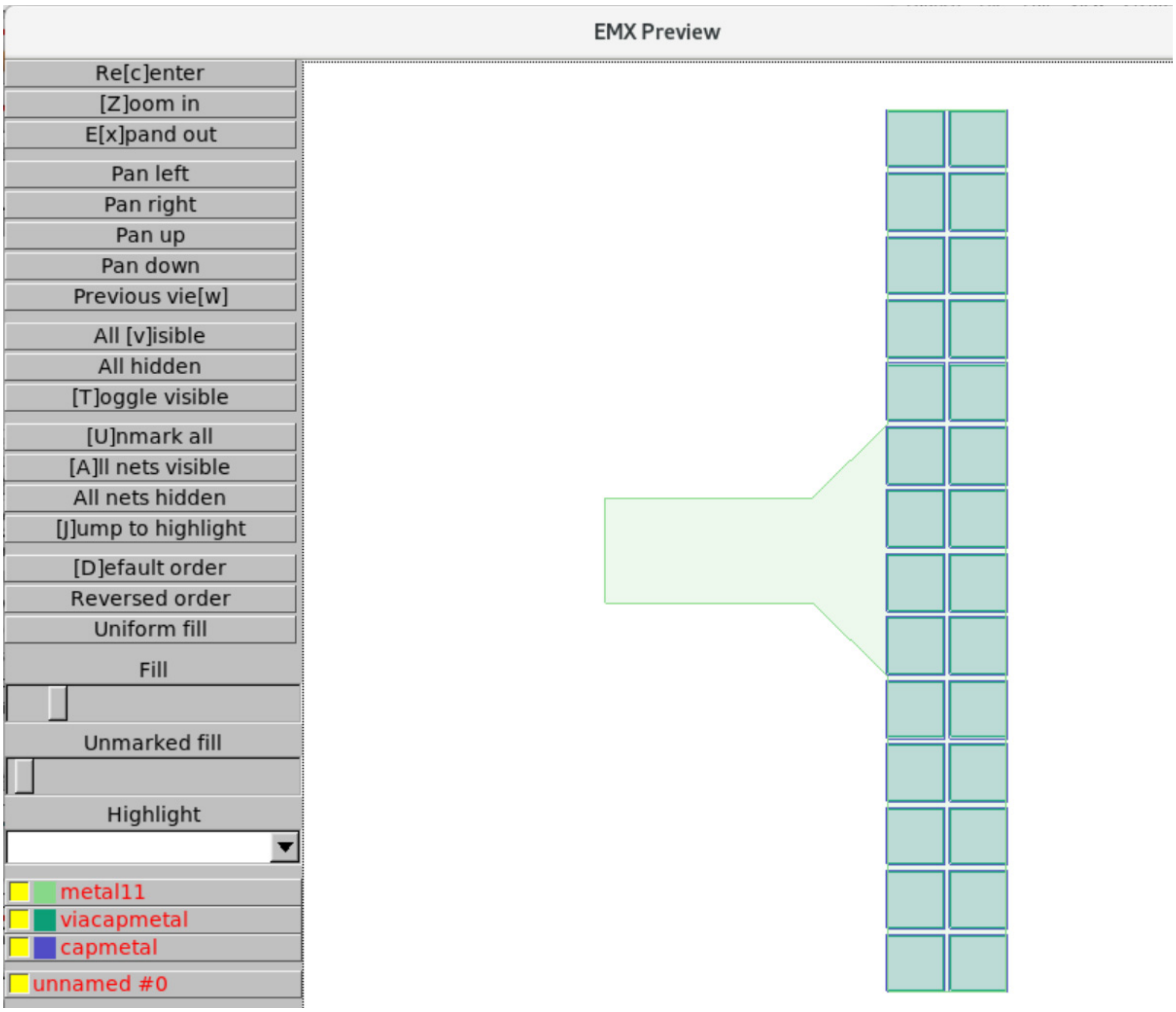
© 2023 Cadence Design Systems, Inc.

LVS or restart the EM Assistant.

3. Add a net to the model. The net or instance selection needs to be made on the corresponding schematic.
4. Preview the model in Virtuoso by clicking *Preview EM Layout* in the *Pre-Process* tab. Alternatively, preview the model in EMX gdsview with *Preview Geometry*.

© 2023 Cadence Design Systems, Inc.

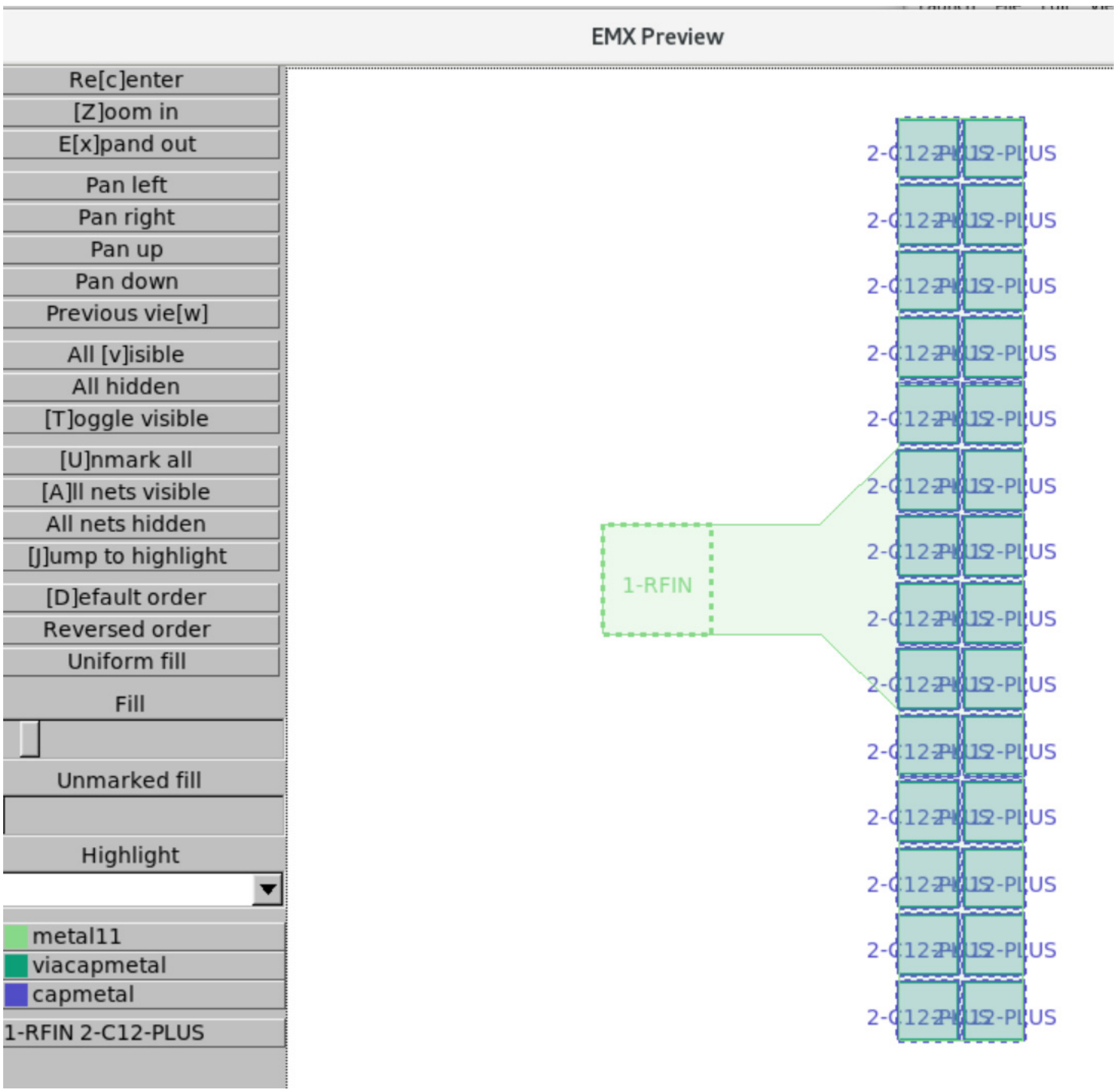
This exports a CLF file and displays it in `gdsview`. The layout is shown exactly as the EMX solver will see it.



5. Generate ports.

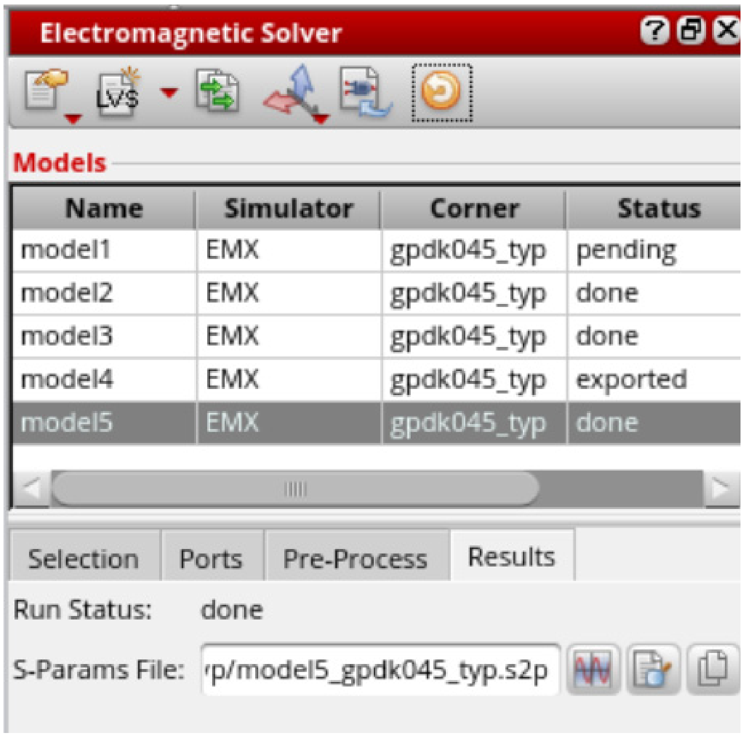
© 2023 Cadence Design Systems, Inc.

6. Run *Preview Geometry* again. You will see the ports and the net names in the EMX preview.



© 2023 Cadence Design Systems, Inc.

7. Run simulation and load the results.

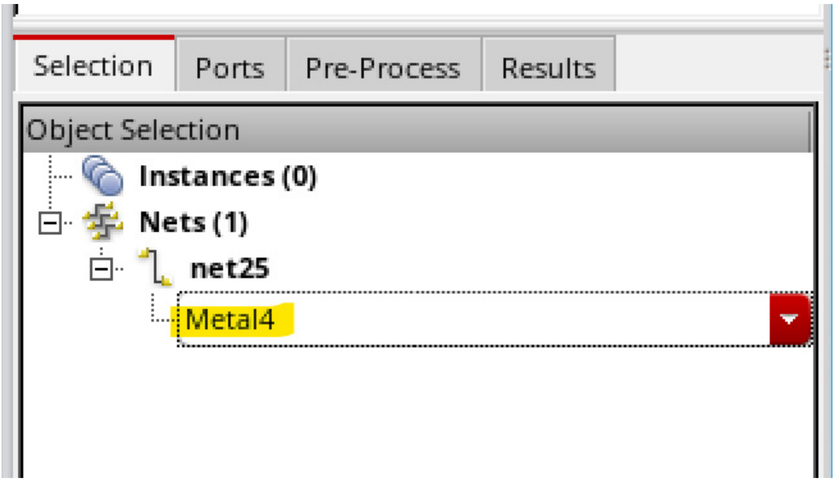


8. Create an extracted view. The reference view for the extracted view must be a Quantus Smart View. In the Create Extracted View form, you can choose Quantus Smart Views only when the model is of type LVS. When stitching, the Smart View's ground net is used as the ground net for the `nport`.

Additionally, you can partially extract a net that connects to a device layer. For partial net extraction, you can expand a net by clicking + and specifying a layer name. This creates an EMX port on every shape of the specified layer. Such ports are reconnected to the Quantus

© 2023 Cadence Design Systems, Inc.

parasitics below the specified layer. Ensure that the *EM Analysis* mode is enabled in Quantus when using partial nets.

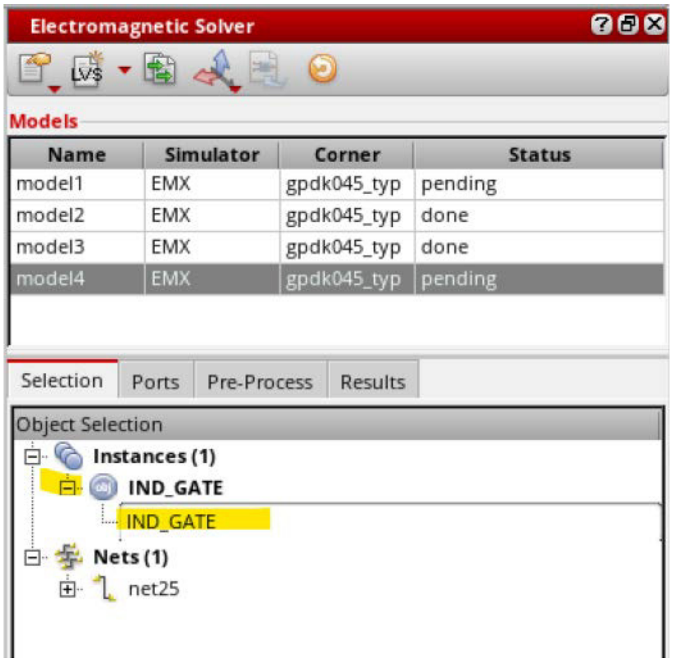


You can extract only the upper layers with EMX and use Quantus RC elements for the lower layers. Many basic PDK cells, such resistor and capacitors, can be extracted by the LVS flow. Add the instances of these cells into an EMX model.

The instances of LVS blackboxed cells can also be added to the EMX models. To include instances that are blackboxed in the LVS flow, expand the instance name and specify the name of the corresponding layout instance. The content of blackboxed instances is not read

© 2023 Cadence Design Systems, Inc.

from the SVDB file. The EM Solver assistant reads it from the Virtuoso layout. Consequently, you need to provide the name of the layout instance to use.



Related Topics

Creating Extracted Views from Models

svdbDirectoryLabelText

quantusLayerSetupFile

quantusTrpFile

lvsLayerMapFile

LVS Layer Map File

The layer map file is essential in the LVS flow to map the layers. Refer to the following syntax of the file:

```
(nonEMagLVSLayers
(
    (l_LVSLayerNames) …
  )
```


© 2023 Cadence Design Systems, Inc.

```
)

(layers
  (
    (virtuosoLayerName  l_LVSLayerNames) …
  )
)

(devices
  (
    (lvsRecognitionLayerName   l_virtuosoLPPs)…
  )
)
```

`nonEMagLVSLayers` specifies a list of one or more LVS layer names. It is an error if a shape on such a layer is found in the EM model. For partial nets, any shape on such a layer is assumed to be part of the Quantus portion of the net.

`layers` specifies a list of entries and each entry starts with a `virtuosoLayerName` followed by one or more LVS layer names. All specified LVS layer names are mapped to that virtuoso layer. LVS layers that have the same name as the Virtuoso layer are mapped automatically, for example, if a layer is called `M1` in LVS and Virtuoso, no mapping entry is needed.

`devices` specifies a list of entries and each entry starts with an LVS recognition layer, followed by one or more Virtuoso layer-purpose pairs. This is used when PDK elements, such as resistors or capacitors are included in the model. The device is identified using the specified LVS recognition layer and all shapes on the specified Virtuoso layer-purpose pairs that overlap the recognition shape are sent to EMX.

For metal resistors, only a single Virtuoso layer-purpose pair is specified. For more complex devices, such as inductors, multiple layer-purpose pairs might be used.

Here is an example for LVS layer map file:

```
(layers
  (
    ("Cont"   "cont_pdiff" "cont_ndiff" "cont_poly")
    ("Poly"   "poly_conn")
    ("Metal1" "metal1_conn")

    ("Via10" "via10_cap" "via10_nocap")
  )
)
```


© 2023 Cadence Design Systems, Inc.

```
(devices
  (
    ("_resm2" ("Metal2" "drawing"))
    ("_resm5" ("Metal5" "drawing"))
    ("_resm11" ("Metal11" "drawing"))
  )
)
```

The three LVS layers `cont_pdiff`, `conf_ndiff`, and `cont_poly` are all imported into the Virtuoso layer `Cont`. The `poly_conn` layer in LVS is imported to the Virtuoso layer `Poly`.

No mapping is specified for `Via1` because it is called the same in LVS and Virtuoso.

The `Metal2` resistor in LVS is using recognition layer `_resm2`. The LPP `Metal2/drawing` is specified. This means when a `Metal2` resistor is added to the model, the `Metal2/drawing` shape within the recognition area is sent to EMX.

Related Topics

Extracting Nets with EMX Using the LVS Flow

quantusLayerSetupFile

quantusTrpFile

lvsLayerMapFile

© 2023 Cadence Design Systems, Inc.

Creating S-Parameter Cellviews for Models

In addition to saving S-Parameters as extracted views, you can save them as S-parameter cellviews along with their symbols that can be instantiated in a schematic cellview.

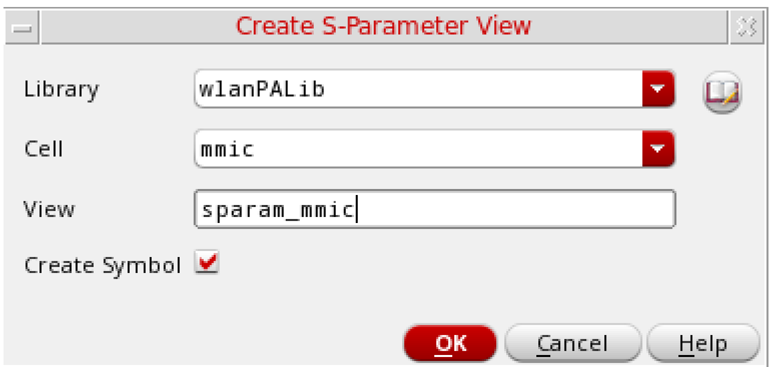
To save a model as an S-parameter cellview:

1. After the simulation for the model is complete, refresh its status.

   The value in the *Status* column for the model is shown as *done*.

2. Right-click the row for the model and choose *Export to S-Parameter View*.

   The Create S-Parameter View form is displayed with a default name to be used for the new S-parameter view to be saved in the current library and cell. However, you can customize the *Library*, *Cell*, and *View* names.



3. (Optional) Click the button to the right of the *Library* drop-down list to open the Library Browser form and choose another library or cell.

4. (Optional) Change the name of the view.

5. Click *OK*.

   A new S-parameter view and a symbol view is saved in the specified library and cell.

Related Topics

Create S-Parameter View Form

exportCellName

exportLibName

exportSparamViewName

© 2023 Cadence Design Systems, Inc.

Extracting S-Parameter Models from Full Cellviews (IC Layout Only)

There can be scenarios where you need to extract a model for the complete layout or a major part of it. In both cases, instead of manually selecting a large number of nets or instances, you can use the *Extract Full Cellview* feature of Electromagnetic Solver assistant.

Related Topics

Creating an S-Parameter Model for the Complete Layout (Passive Devices)

Creating an S-Parameter Model for a Layout-Driven Flow

© 2023 Cadence Design Systems, Inc.

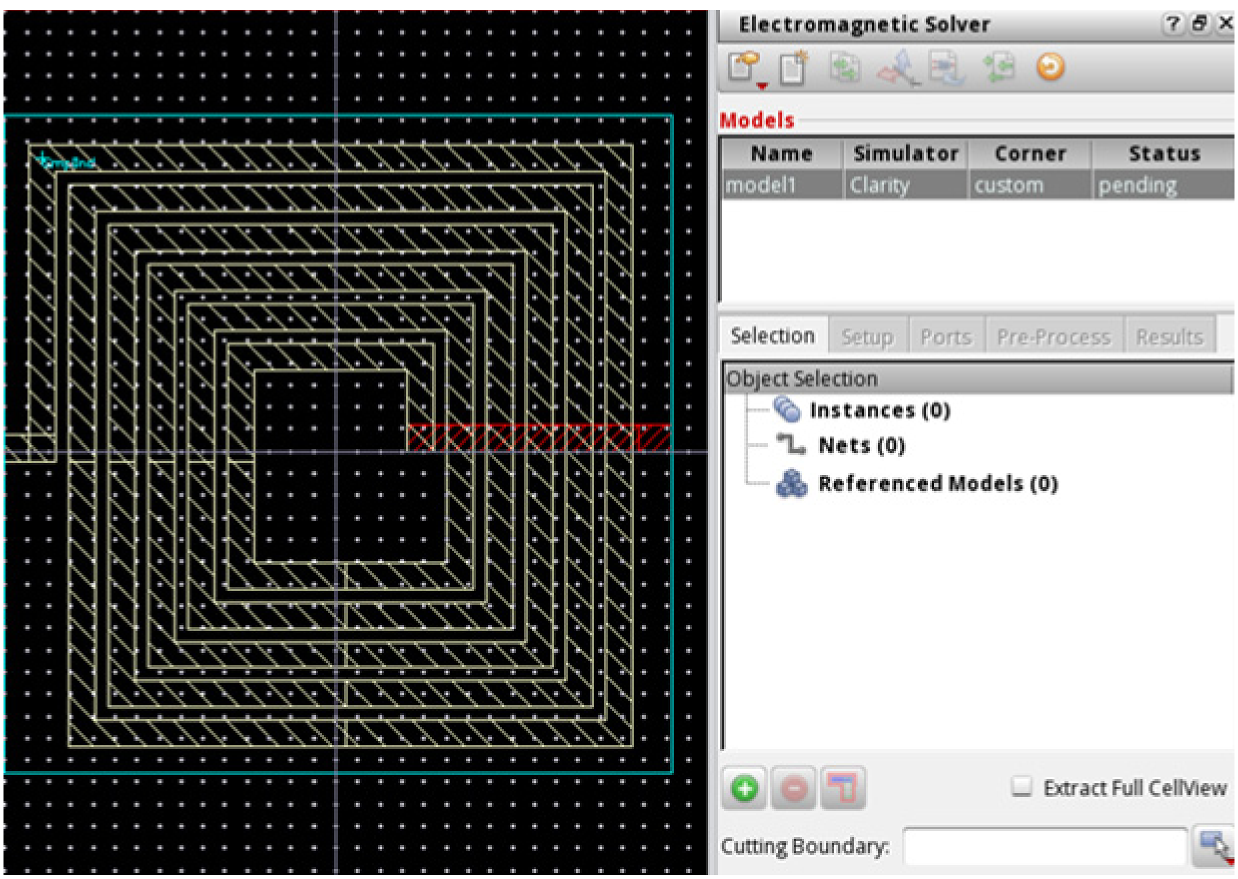
Creating an S-Parameter Model for the Complete Layout (Passive Devices)

If the IC layout is a passive device, we can characterize that layout cellview as an S-parameter cellview. Similarly, in scenarios where an IC layout contains devices, such as inductors or MIMCAP devices, but you cannot access the nets and connectivity details, you can export the model for the complete layout as an S-parameter cellview, which is an S-parameter results file. In addition, you can create a symbol view, which you can use to instantiate the model in a schematic.

To extract the model for a layout as a complete S-parameter cellview:

1. Open the layout in Layout MXL and set the workspace to `Electromagnetic`.

   The following sample layout contains an inductor for which connectivity details are not available:

2. Specify the process setup in the process corner directory.
3. Create a new model in the Electromagnetic Solver assistant.

© 2023 Cadence Design Systems, Inc.

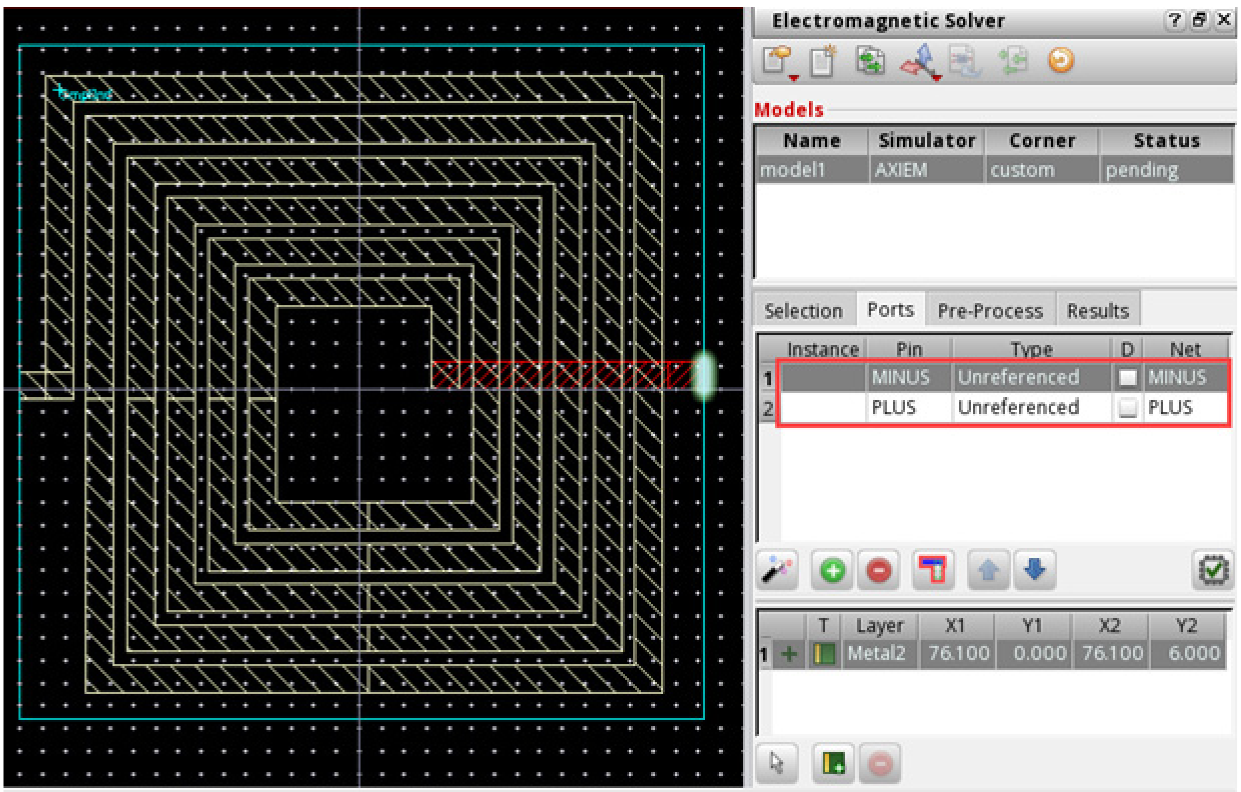
4. Select a process corner and specify a simulator that you want to use to run electromagnetic simulation.

5. Select the *Extract Full Cellview* check box at the bottom of the Electromagnetic Solver assistant.

   Note: The objective is to extract the full cellview. Therefore, it is not required to select any specific instance, net, or referenced model. Notice that the object selection controls on the *Selection* tab are disabled after you select the *Extract Full Cellview* check box.

6. On the *Ports* tab, click *Automatically Generate Ports* to create ports for the top-level pins of the instances available in the layout.

   For the example layout shown above, the tool automatically identifies the two top-level pins, PLUS and MINUS, and adds those to the ports table, as shown below.



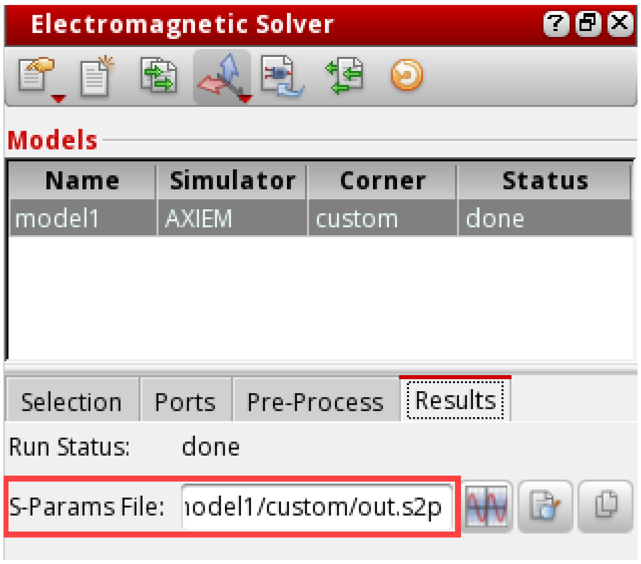
   Similarly, if the layout contains two inductors, the tool identifies ports of both inductors and adds four ports to the port table.

   Note: For Clarity simulators, the *Automatically Generate Ports* command is disabled because the reference net is not selected. In that case, you might need to manually add ports by using the *Add Port* command and then adjust the placement of the pins.

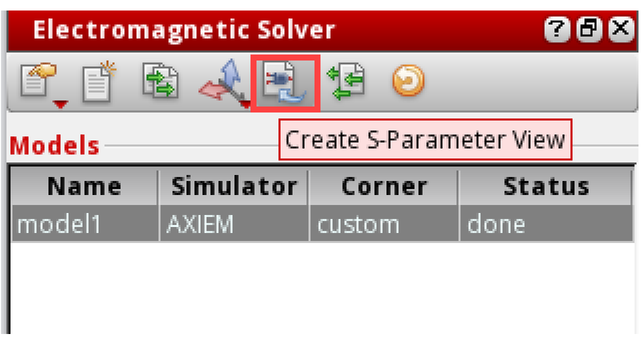
© 2023 Cadence Design Systems, Inc.

7. Run the simulation by using an appropriate command for your selected simulator.

   After the simulation run is complete, the path to the S-parameter file is available on the *Results* tab of the Electromagnetic Solver assistant.

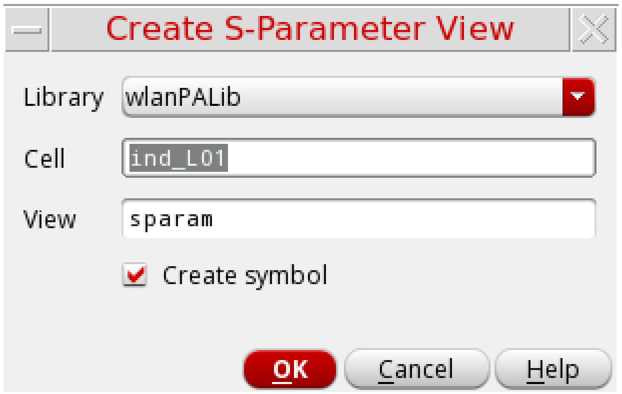


8. Click Create *S-Parameter View* on the toolbar of the Electromagnetic Solver assistant.



© 2023 Cadence Design Systems, Inc.

The Create S-Parameter View form appears.



9. Specify a name for the cell and S-parameter view in the *Cell* and *View* fields, respectively. The default view name is `sparam`.

   By default, the *Create Symbol* check box is selected to create a symbol view too. Deselect it if you do not want to create a symbol.

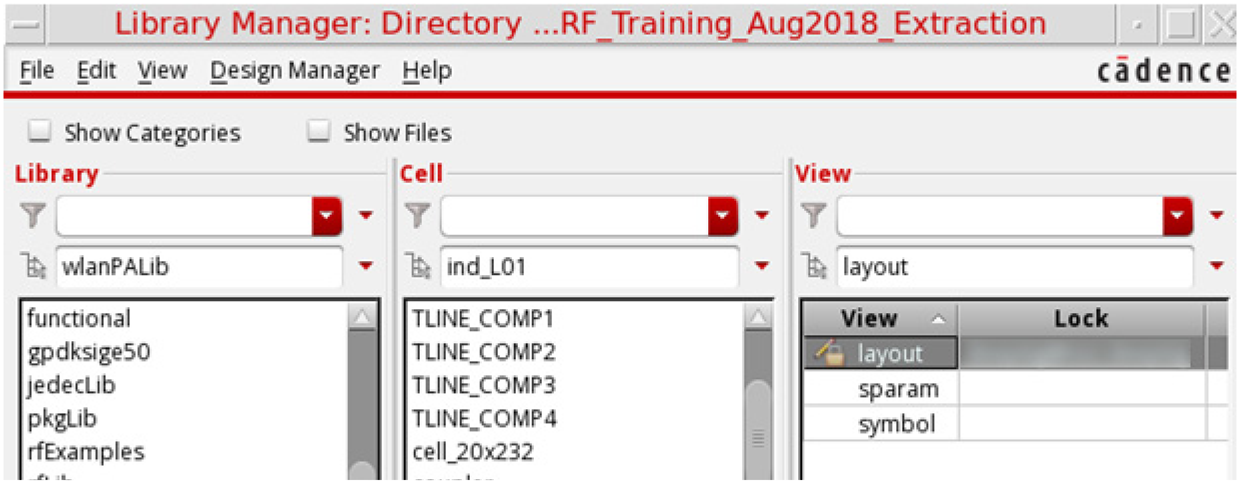
10. Click *OK* to create the cellviews.

    The CIW shows messages indicating that the new cellview has been created.

```
Loading schematic.cxt
Loading ddui.cxt
Symbol (ind_L01 symbol) generated and saved in library:wlanPALib.
Loading cdf.cxt
INFO (SPARAMVIEW-7075): Cellview 'wlanPALib/ind_L01/sparam' created.
```


© 2023 Cadence Design Systems, Inc.

The model has now been successfully extracted as a complete cellview. The new sparam view and the symbol view are visible in the Library Manager. These are also added to the library database.



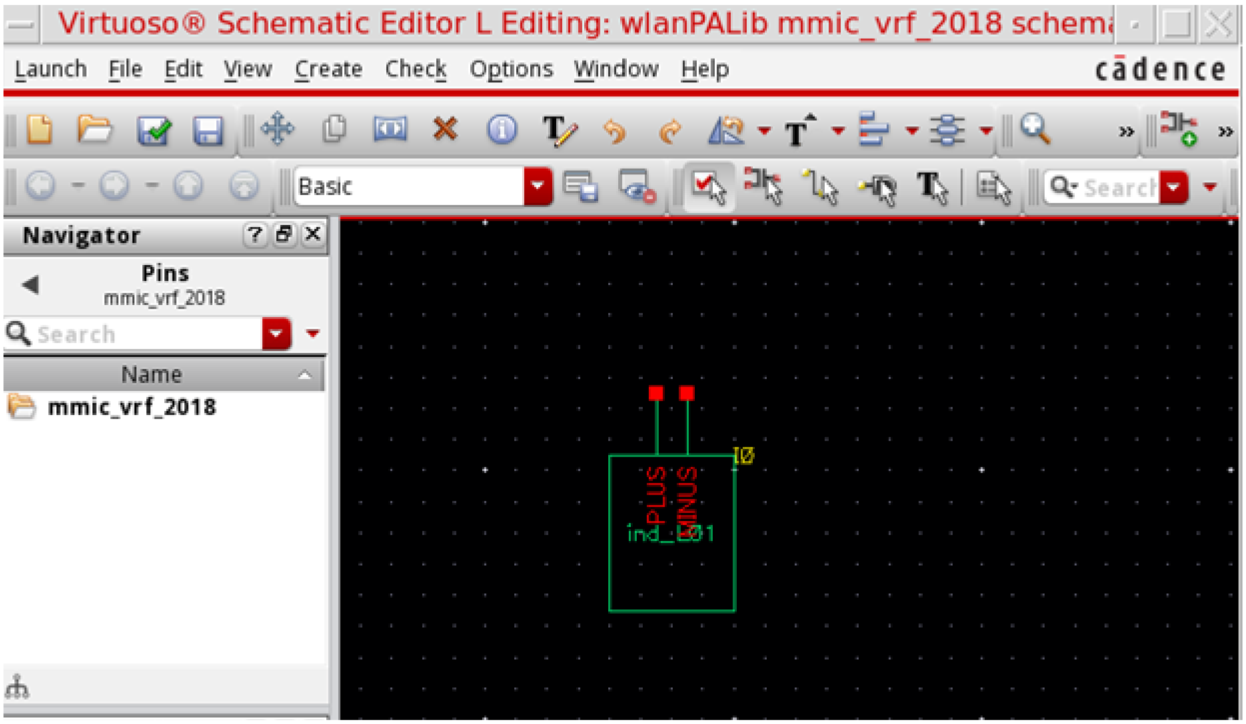
You can double-click the sparam cellview to open and review its contents. The sparam views are text-based and are opened in the default text editor.

```
! GNDNET
! NETSLIST
! Port 1 = MINUS
! Port 2 = PLUS
!ENCODED_BEGIN
!0582B71428CDB13794A3D07085865A4D22A0CF4145B0166C94D538E66A75298A693F4EAC31C54FC11303758BD538
760F0E51D196C8CFA21399C1743E0E8B54CF6C
!ENCODED_END
! AWR AXIEM (9293)
! Sun Jan 12 23:26:30 2020
! nPorts: 2, nFreqs: 1001

# HZ S RI R 50
0 0.0626732957414255 -0 0.937326694821046 -0 0.937326694821046 -0 0.0626732964710386 -0
10000000 0.062676018334936 3 0.00148787670989173 0.937323340052759 -0.00181225590775418 0.9373
23340052758 -0.00181225590775418 0.0626760776656 07 0.00148476866775723
20000000 0.0626841860187942 0.002975725280379 0.937313275891084 -0.00362447456852797 0.937313
275891084 -0.00362447456852 8 0.062684421153173 0.00296950901363551
30000000 0.0626977985120168 0.00446351783517377 0.937296502756765 -0.00543661876423504 0.9372
96502756764 -0.00543661876423 5 0.0626983266562928 0.00445419279787117
40000000 0.0627168553449642 0.00595122650111421 0.937273021352264 -0.00724865128221608 0.9372
73021352264 -0.00724865128221598 0.0627177937108425 0.00593879178362394
50000000 0.0627413558602224 0.00743882340970685 0.937242832660895 -0.00906053491794912 0.9372
```


© 2023 Cadence Design Systems, Inc.

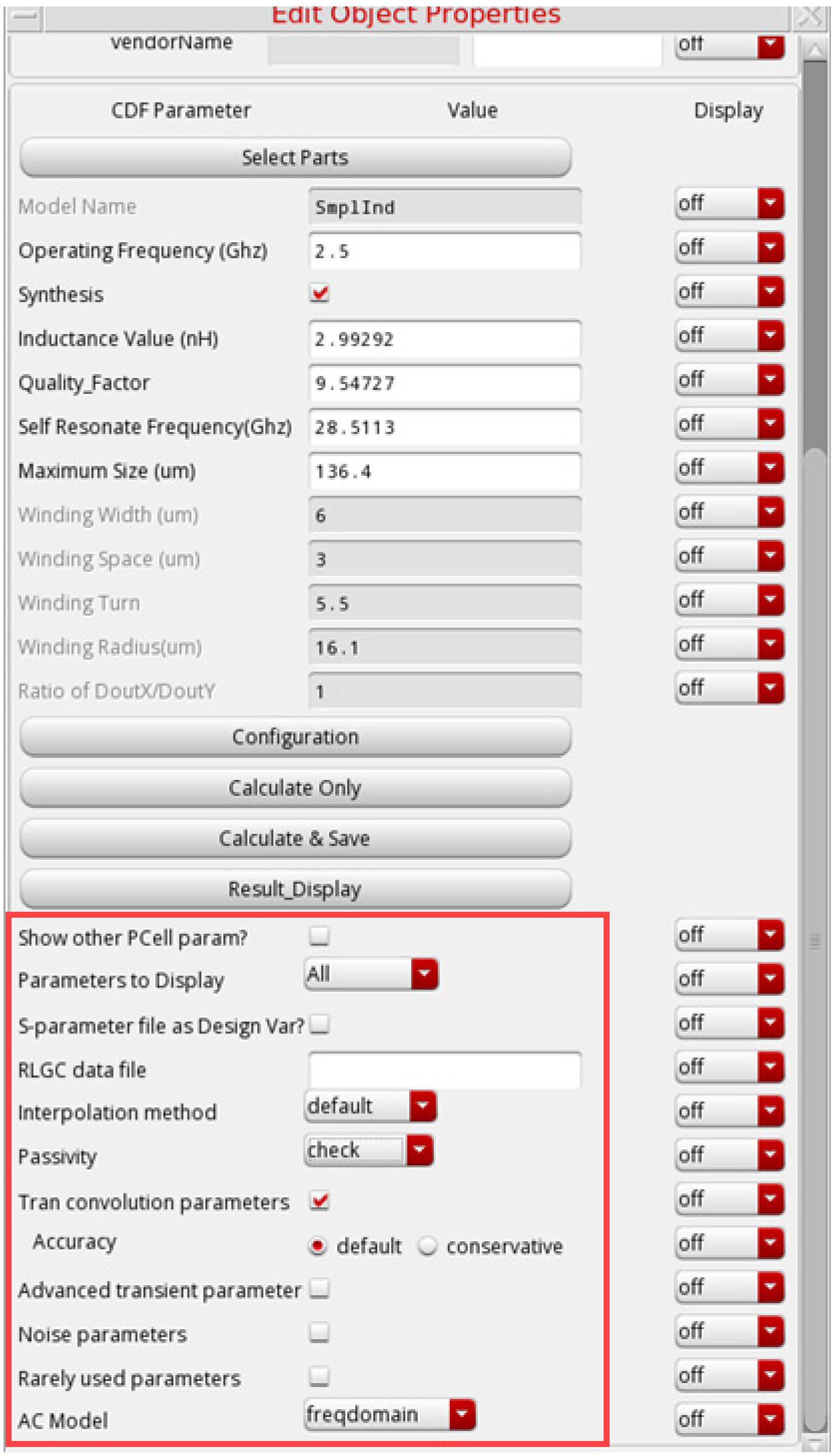
You can use the extracted S-Parameter cellviews by creating their instances in a schematic view, as shown in the following figure:



The pins of the instances can be used to establish connections with other devices in the schematic to build the design.

© 2023 Cadence Design Systems, Inc.

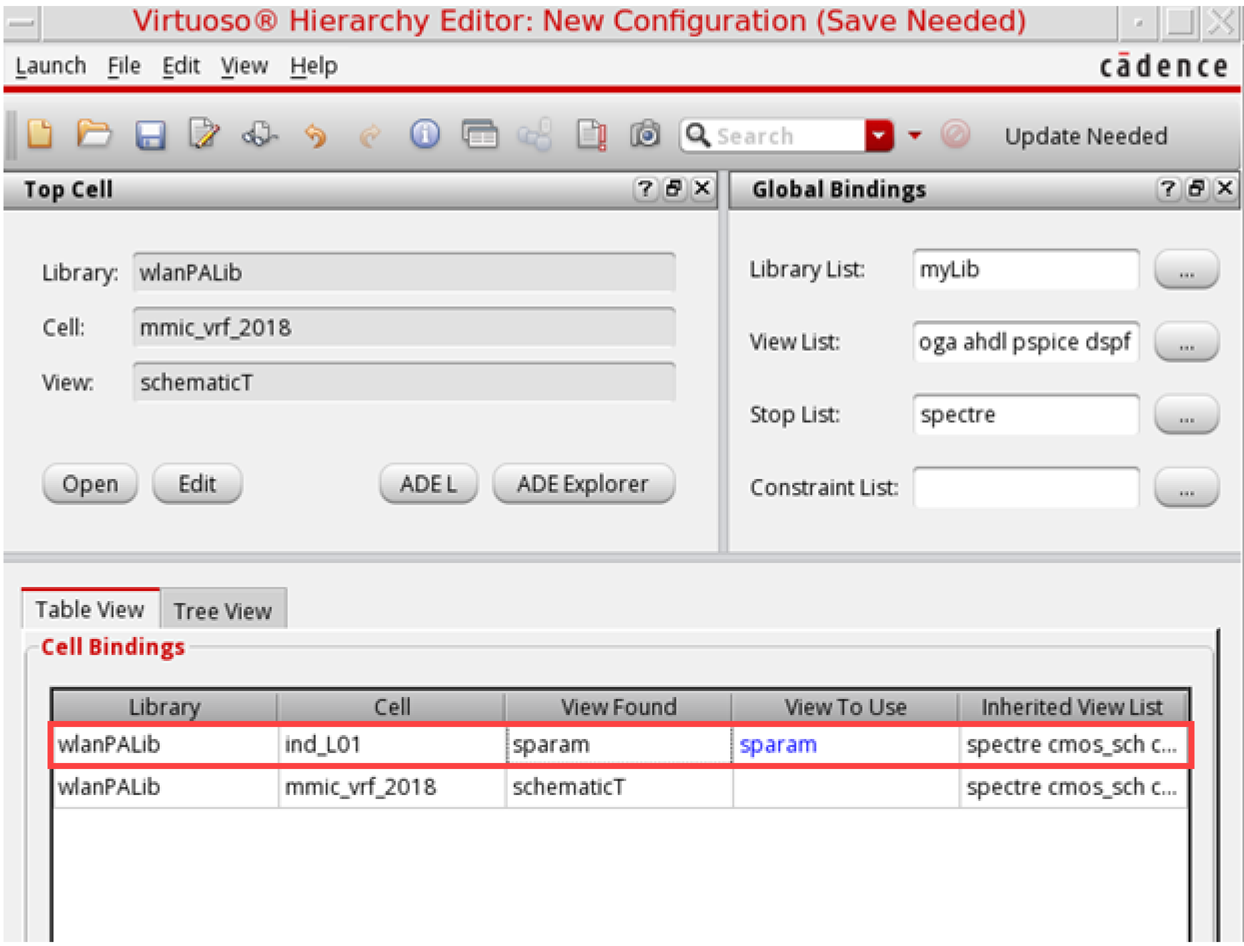
You can right-click the instance in the schematic and choose *Properties* to explore the properties of this instance. The properties listed in the Edit Object Properties form are similar to any other n-port object.



© 2023 Cadence Design Systems, Inc.

Before netlisting a design that contains an instance of your sparam view, provide a mapping in the config view because the schematic view of this object is not available.

The following example shows how to bind an sparam view in the Virtuoso Hierarchy Editor.



For more details about cellview binding, refer to Defining Bind-to-Open on a Per Cell Basis in *Virtuoso Hierarchy Editor User Guide*.

For details about creating a netlist and running simulations, refer to *Virtuoso ADE Explorer User Guide*.

© 2023 Cadence Design Systems, Inc.

Creating an S-Parameter Model for a Layout-Driven Flow

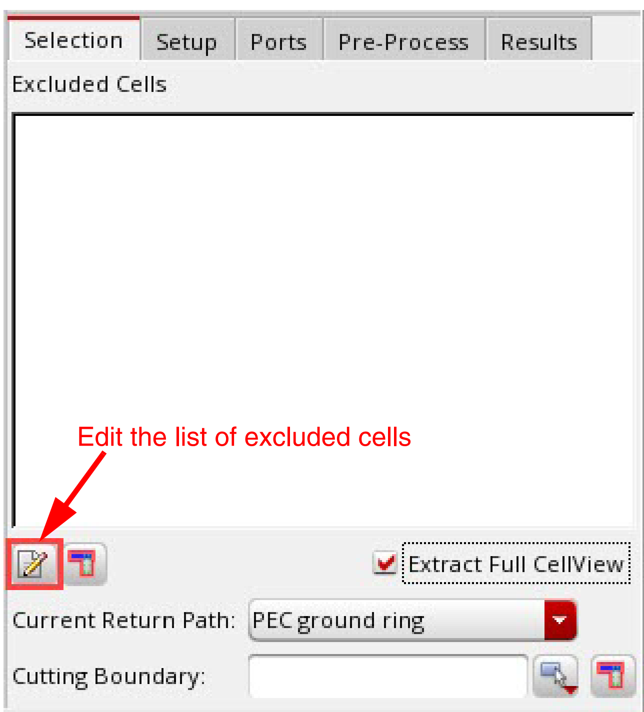
When you do not have a schematic or the connectivity information in the schematic is incomplete, you can run the layout-driven flow in which the entire information is taken from the layout. After the model is simulated, you can use the S-parameter file to create a schematic view that you can use to run simulations.

To create models in this scenario, you can use the full cellview and exclude the instances of active cells, such as MOSFET, NFET, PFET, or BJT. When you exclude these instances, automatic port generation creates ports on all the pins of the excluded cells.

To create a model for a layout-driven flow:

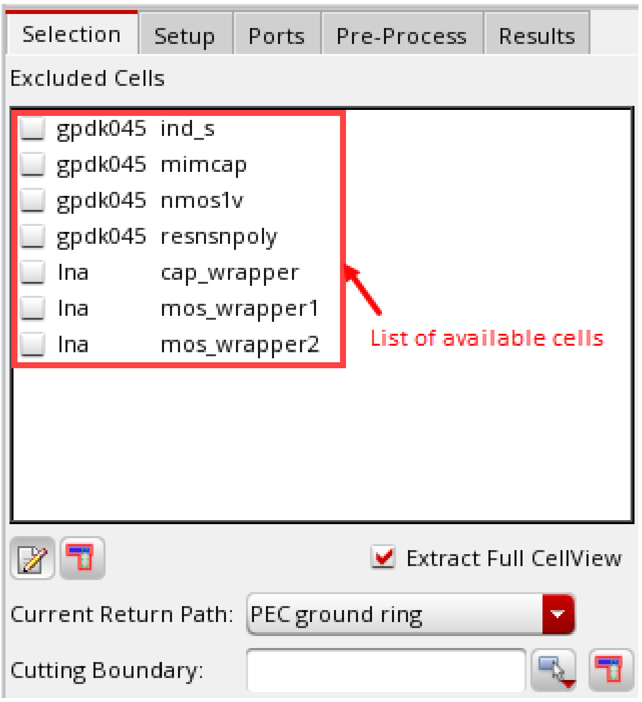
1. Open the layout view in Virtuoso Layout MXL and set the workspace to `Electromagnetic`.
2. Select *Extract Full Cellview* in the Electromagnetic Solver assistant.

   The default list of instances, nets, and referenced models is removed from the *Selection* tab.
3. Click *Edit the list of excluded cells*.



© 2023 Cadence Design Systems, Inc.

The tool looks for the cells that have a symbol view and displays their names in the *Excluded Cells* list.



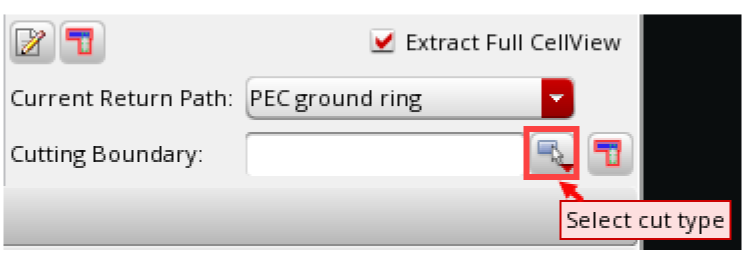
4. Select the check boxes to the left of the cell names that you need to exclude from the model.

   Alternatively, press `Ctrl` and click the names of all the cells to be excluded. Next, right-click and choose *Check*.

5. After the selection of the cells to be excluded is complete, click anywhere out of this list to complete editing of the list.

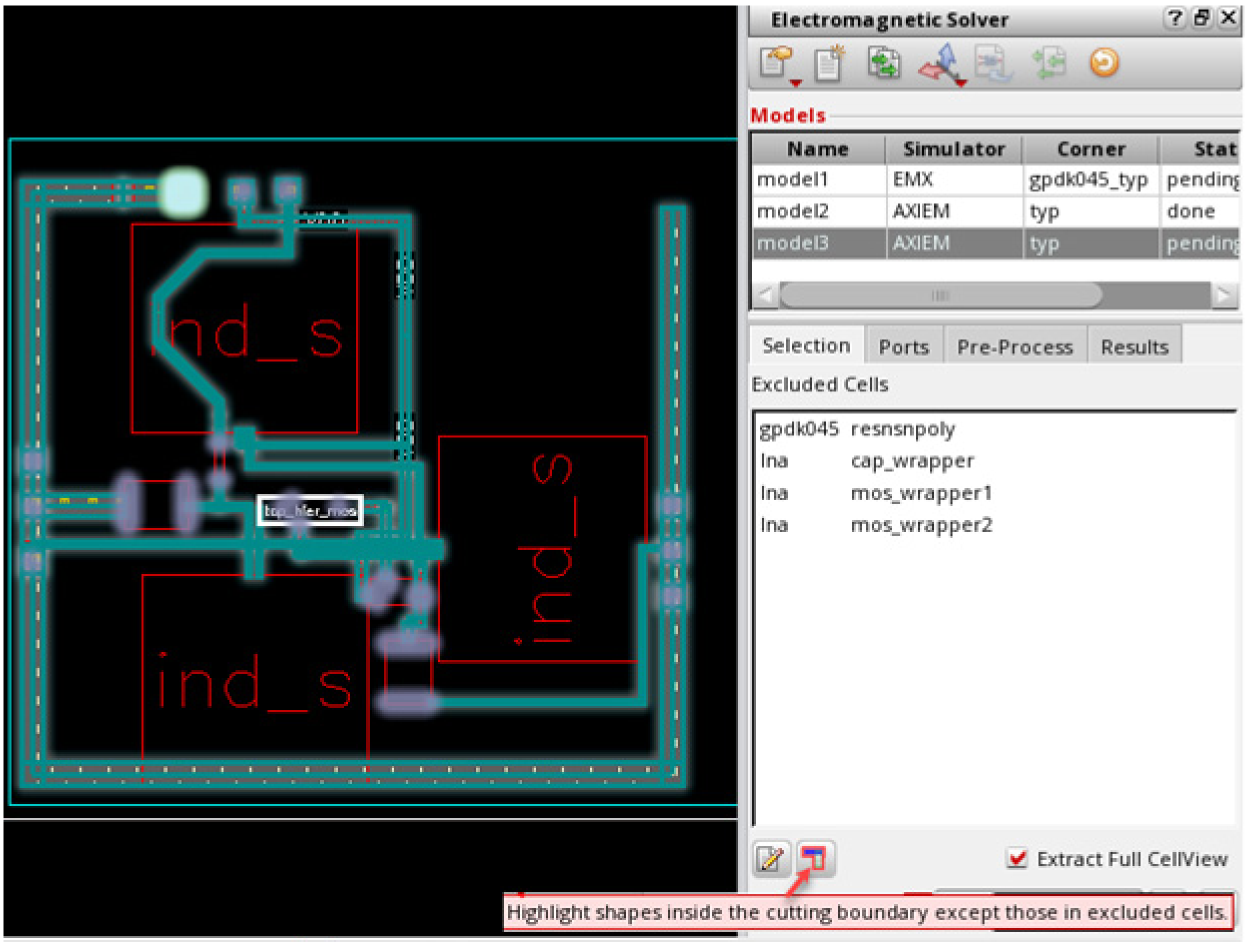
   The *Excluded Cells* list shows all the cells you selected.

6. (Optional) Click *Select cut type* to choose a shape for the cutting boundary and create a boundary to define the area to be used in the model.



© 2023 Cadence Design Systems, Inc.

7. Click *Highlight shapes inside the cutting boundary except those in excluded cells* and verify the shapes on the layout canvas.

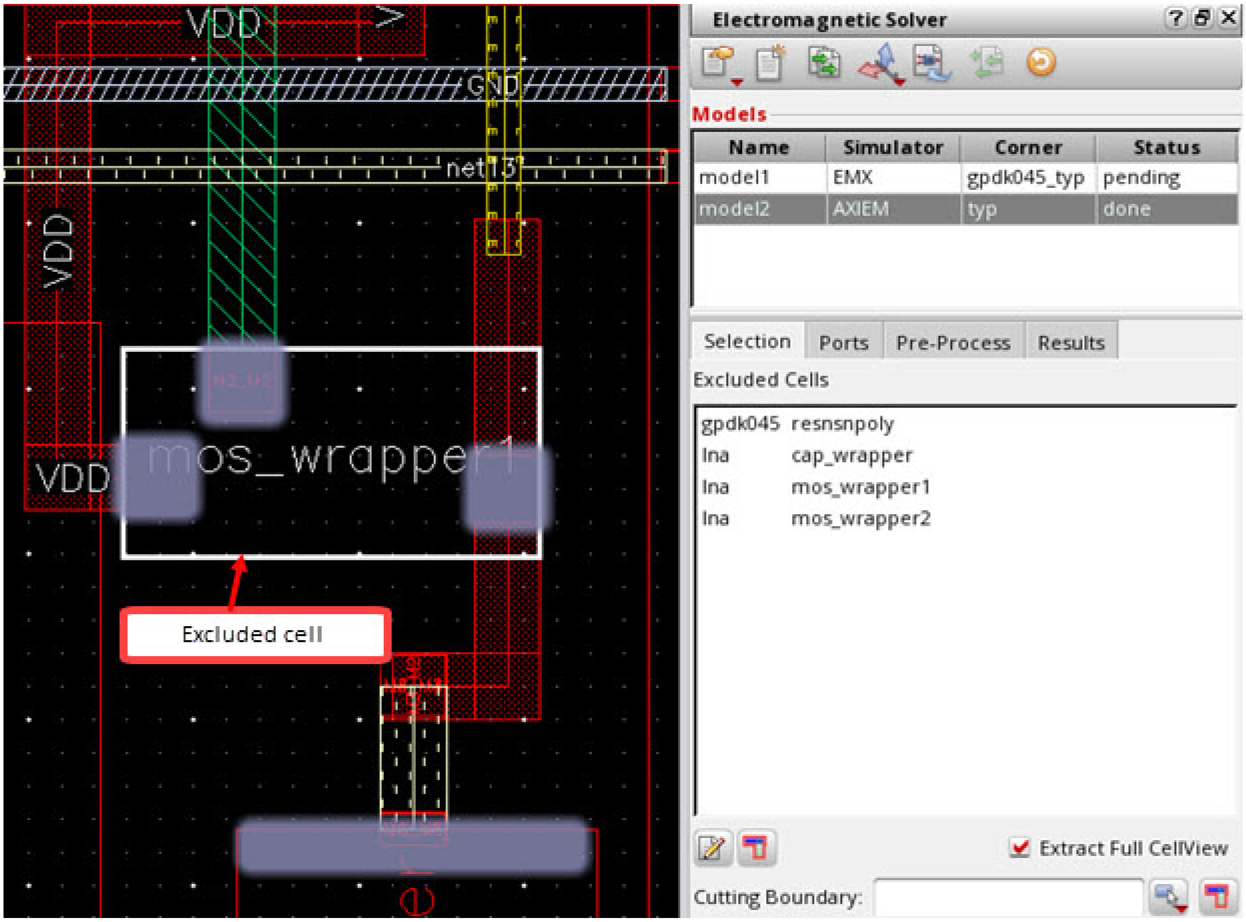


8. Click *Automatically generate ports* on the *Ports* tab to generate ports for the cells in the model.

   The tool uses the general rules to create ports for the included devices.

© 2023 Cadence Design Systems, Inc.

For excluded cells, it creates a port at each pin of an excluded cell. In the example below, ports are created at all three pins of `mos_wrapper1`.

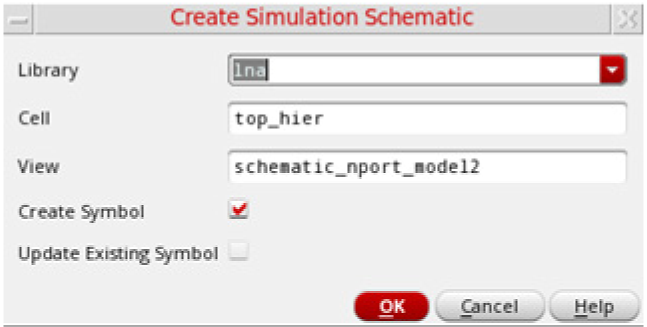


9. (Optional) Specify options on the *Pre-Process* tab to run shape simplification.
10. Generate and view mesh.
11. Run simulation for the model.

    After the simulation is run successfully, the status of the model is changed to `done` and the path to the S-parameter file is displayed on the *Results* tab.
12. Click *Create Simulation Schematic* to create a schematic using the S-Parameters.

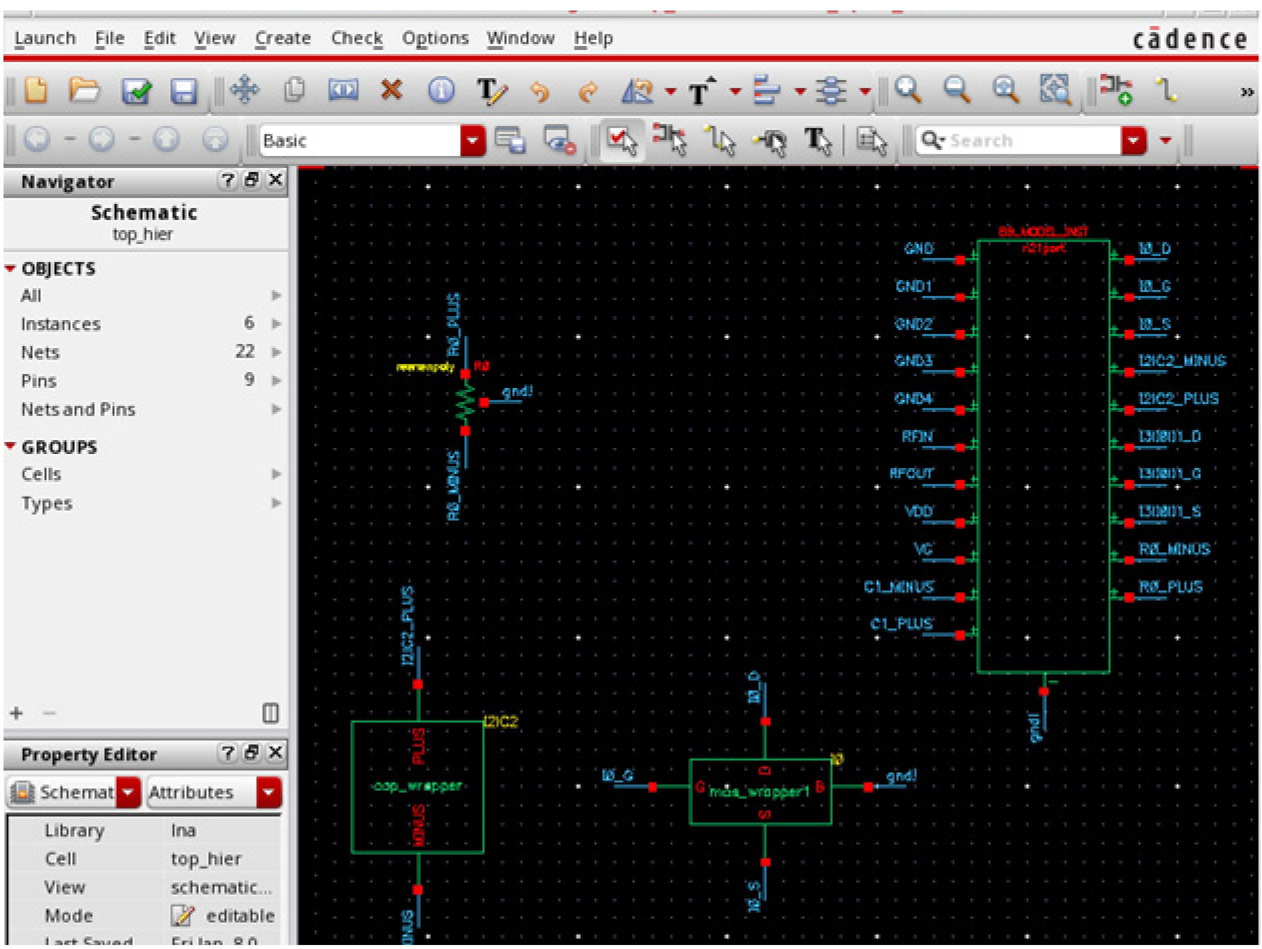
© 2023 Cadence Design Systems, Inc.

The Create Simulation Schematic form is displayed.



13. Specify a name for the new schematic view to be created and click *OK*.

Virtuoso creates a new schematic cellview and adds it to the library. You can open it from the Library Manager.



© 2023 Cadence Design Systems, Inc.

This schematic contains an instance of the S-parameter port and symbols for the excluded cells. The schematic pins are created only for the layout pins that have a port. If common node net exists and is not set to `gnd!`, a pin is created.

Conversely, if a pin on a schematic symbol has no port defined, it is connected to a net that has the `EMX` property set. When a schematic symbol pin has no corresponding S-parameter port, check the EMX string property on the corresponding excluded instance in the layout. The property should be in the format pinName1=netName1 pinName2=netName2. For example, B=GND, which means you can connect pin B of the schematic symbol to the net GND. Once the property is found, check if the pin name is listed in the property. If it is, connect it to the net specified in the property. If there is no property on the instance or the pin is not part of the property, the pin is connected to the same net as the nport common node.

© 2023 Cadence Design Systems, Inc.

Extracting Models for a Cross-Fabric Design

Cross-fabric extraction allows you to create a model across IC, package and board. This lets you capture the full coupling between fabrics in a single 3D simulation. Virtuoso Layout MXL takes care of all the work normally involved in combining IC and package geometry into a merged model.

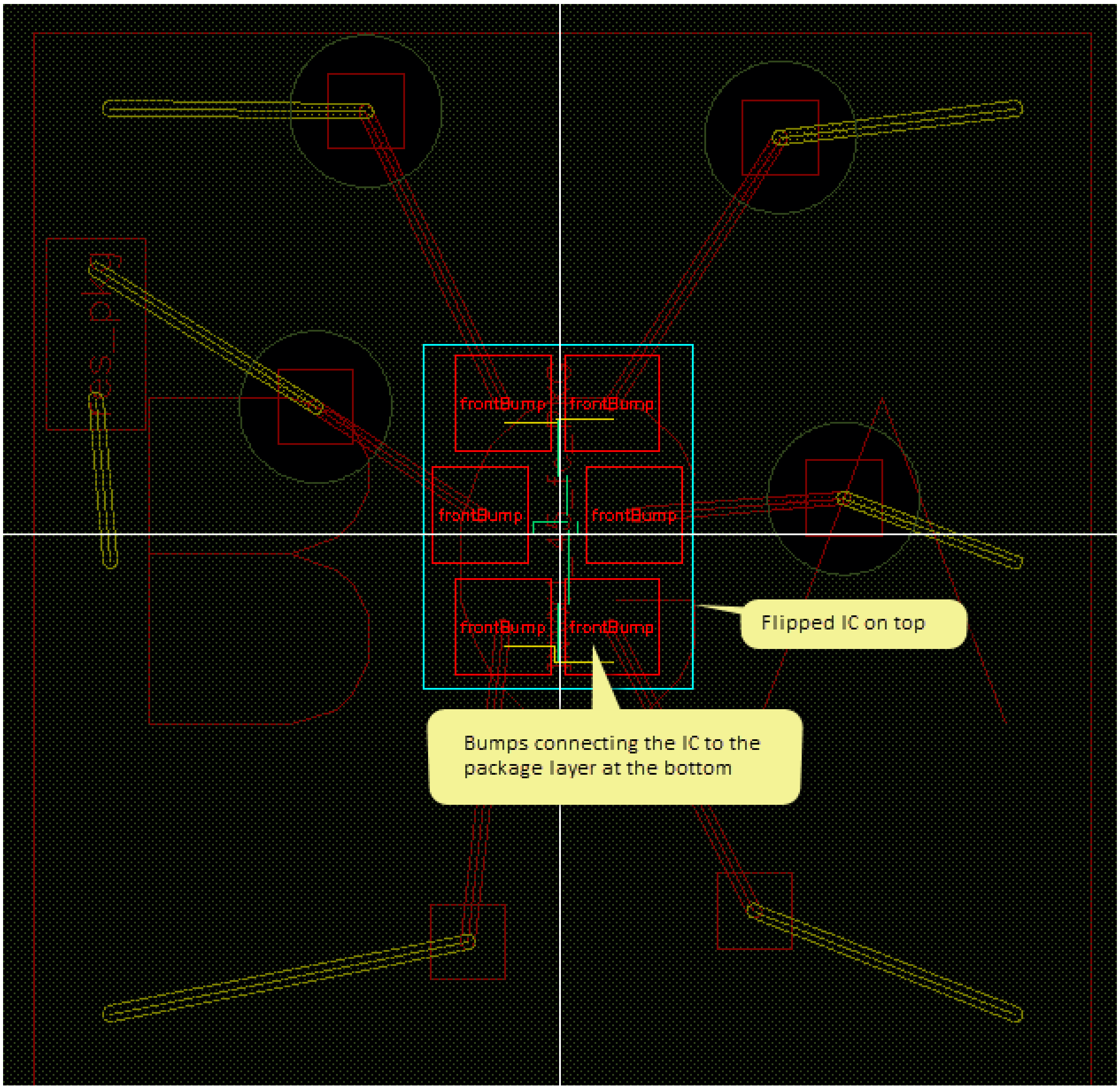
You can read a blog on this feature here.

The Electromagnetic Solver assistant allows you to extract models for a cross-fabric design using Clarity 3D Solver simulator. The objective is to pick objects from different fabrics and generate a common model for them.

A cross-fabric extraction typically runs on a layout cellview that contains at least two different fabrics. Multiple fabrics of either of the IC, package, module, or a board type can be stacked up. Each fabric can have its own technology database and can be connected with other fabrics using balls, bumps, or wirebonds.

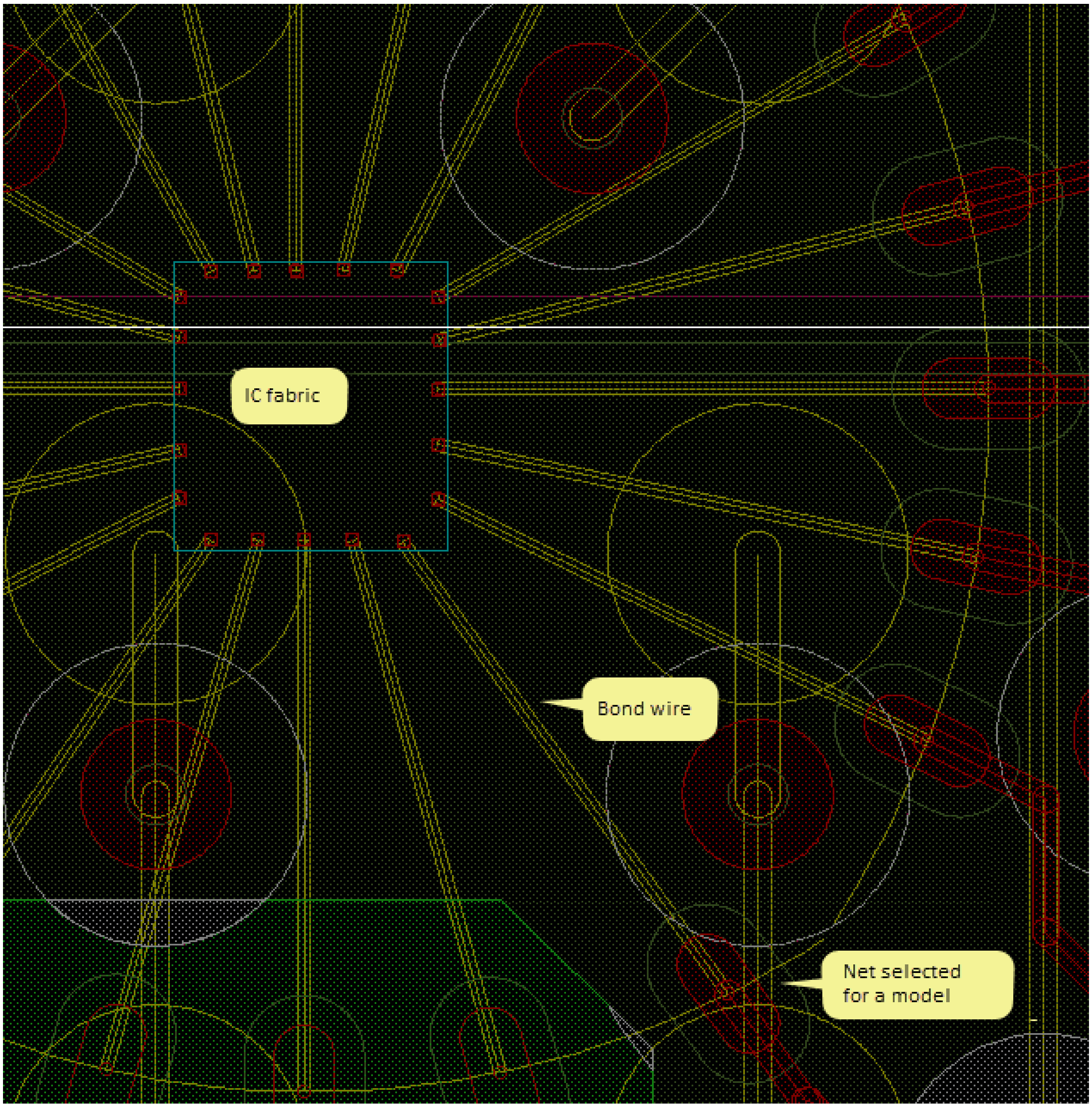
© 2023 Cadence Design Systems, Inc.

The following diagram shows an example of a layer stack containing two fabrics where the IC on top is flipped and connected using bumps:



© 2023 Cadence Design Systems, Inc.

In your designs, you can have ICs that are not flipped and are attached using bond wires, as shown in the following example.



Important

Currently, Virtuoso Layout MXL does not support multiple instances of a fabric placed side-by-side on the outer fabric. This feature works only on a vertical stack of fabrics.

© 2023 Cadence Design Systems, Inc.

For a cross-fabric model extraction in Virtuoso, you begin with the board layout cellview, which is the top fabric or cellview. The board contains an abstract instance of a package TILP Pcell. The package would further contain an abstract instance of an IC layout or another module. There can be various such layers depending on your design.

Each of the package, module, or IC layouts have their own models. For a cross-fabric design, the objective is to extract a model for the outer-most fabric. Each fabric in the stack would reference the models of other fabrics placed within it. For example, the package would reference the model for the IC instantiated in it.

Important

For cross-fabric extraction, it is required that the top-level fabrics in the design refer the models of the lower-level fabrics instantiated in them. Therefore, it is essential to follow the **bottom-up approach for model creation**. For example, create a model for an IC, which is referenced in the model for a package, which is further referenced in the model for a board.

If you have a layer stack ready for extraction, ensure that the following prerequisites are met:

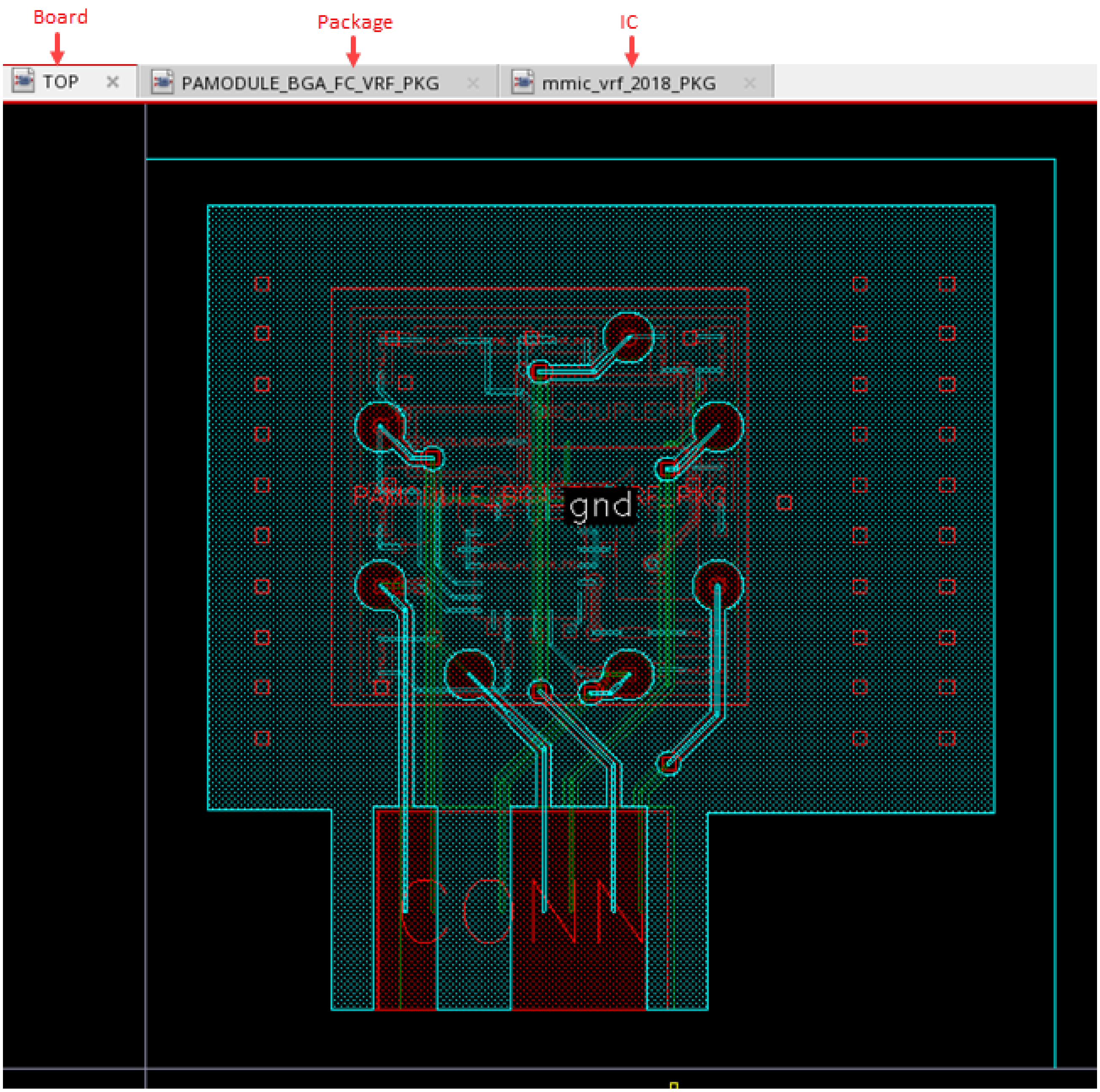
- Each package and board fabric has a dielectric layer on top and bottom. If not already created, you can use the Layer Stack Editor command in the *RF-Module* menu to add dielectric layers.
- The height of the balls and bumps for each abstract instance equal to or higher than the height of the solder mask. You can use the Ball and Bump Editor command in the *RF-Module* menu to modify the dimensions.

Next, you can proceed with the steps given below to run model extraction for a cross-fabric design.

1. Open the outer-most fabric in Virtuoso Layout MXL.
2. Choose *Module – Edit-in-Concert* to enable the Co-Design mode where you can view and edit die packages and their corresponding die layouts synchronously.

© 2023 Cadence Design Systems, Inc.

The following example of a sample design shows the board layout along with the package, and die layouts associated with it opened in the Co-Design mode.



3. Identify the objects you want to include in the cross-fabric model. These objects could include one or more nets, or instances from this layout.
4. If required, create a cutting boundary to define the area to be considered for this fabric.

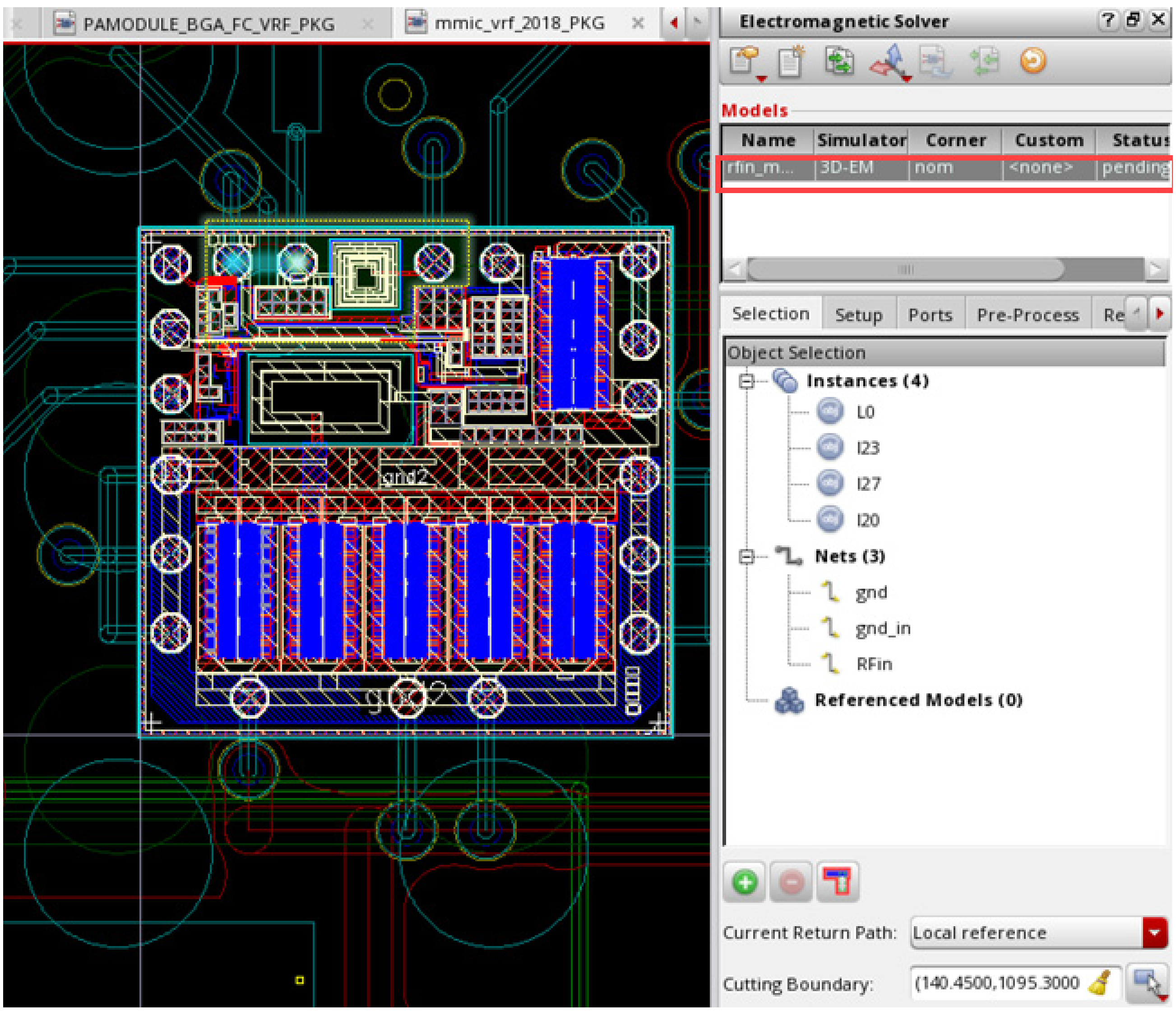
© 2023 Cadence Design Systems, Inc.

You can create cutting boundaries at any level in a cross-fabric model. The cutting boundary for any level defines the region to be used for the fabric at that level. However, the cutting boundary defined for the top-most fabric again cuts through all the fabrics in the design.

5. Following the bottom-up approach, open the layout of the lowest fabric and define a model using the required objects from this layout.

   You can use the select one or more objects on the *Selection* tab to highlight all the objects on different fabrics.

   For the example design shown above, if you select an inductor, two bumps, one MIMCAP, and three nets to be included in a model, the setup appears as shown below.



© 2023 Cadence Design Systems, Inc.

6. Ensure that you choose the Clarity 3D Solver simulator for model generation.

   Only Clarity 3D models with the *Current Return Path* set as `Local Reference` can be referenced from other models.

7. Create and validate ports.

8. Focus on the layout view for the fabric above the previous one and do the following:

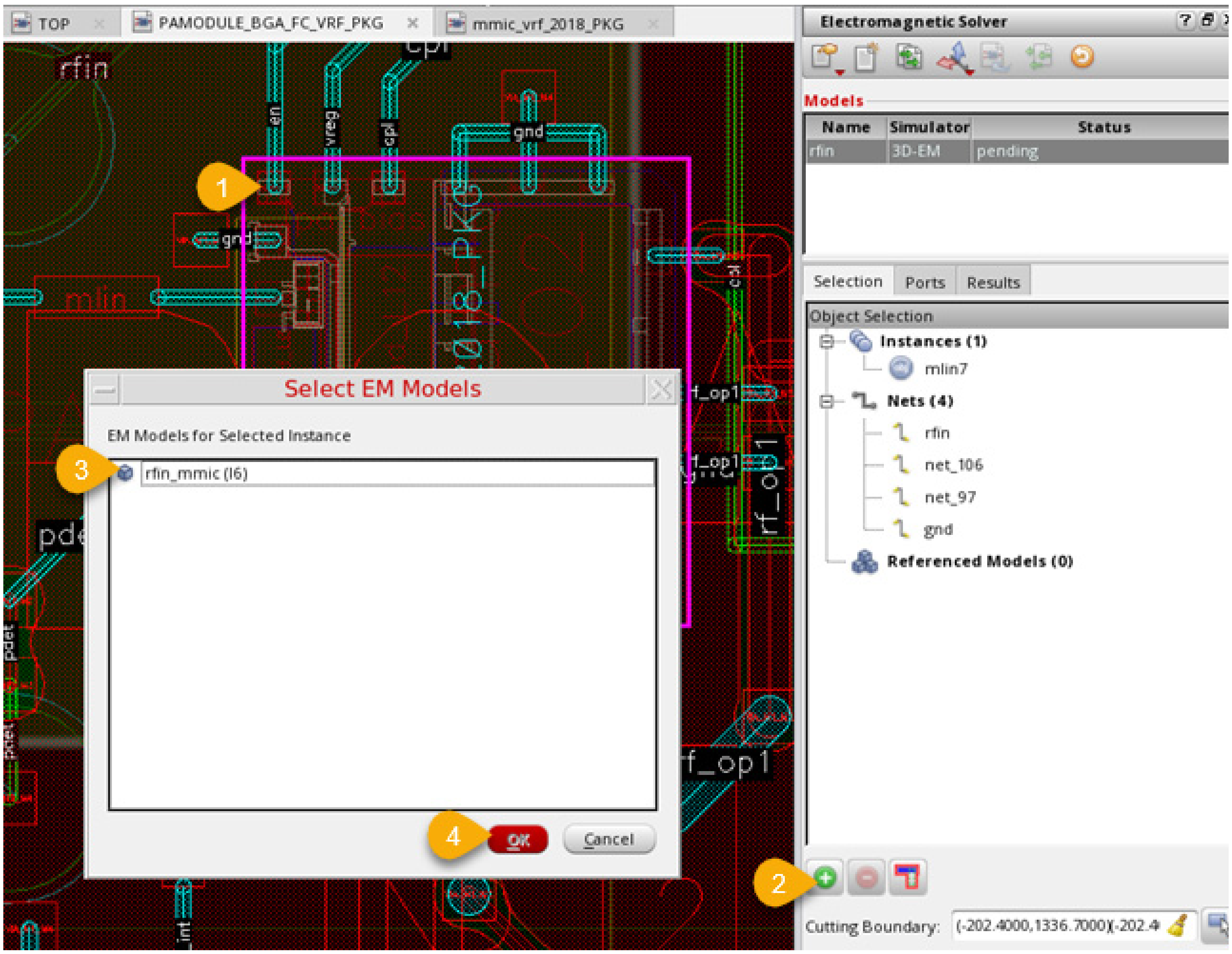
   a. Create a new model and choose the `Clarity` simulator.

   b. Select objects to be included from this layout.

   c. Click the die abstract and click on the *Selection* tab of Electromagnetic Solver assistant.

© 2023 Cadence Design Systems, Inc.

The Select EM Models form is displayed. All the models that can be referenced for the selected die abstract are listed in this form.



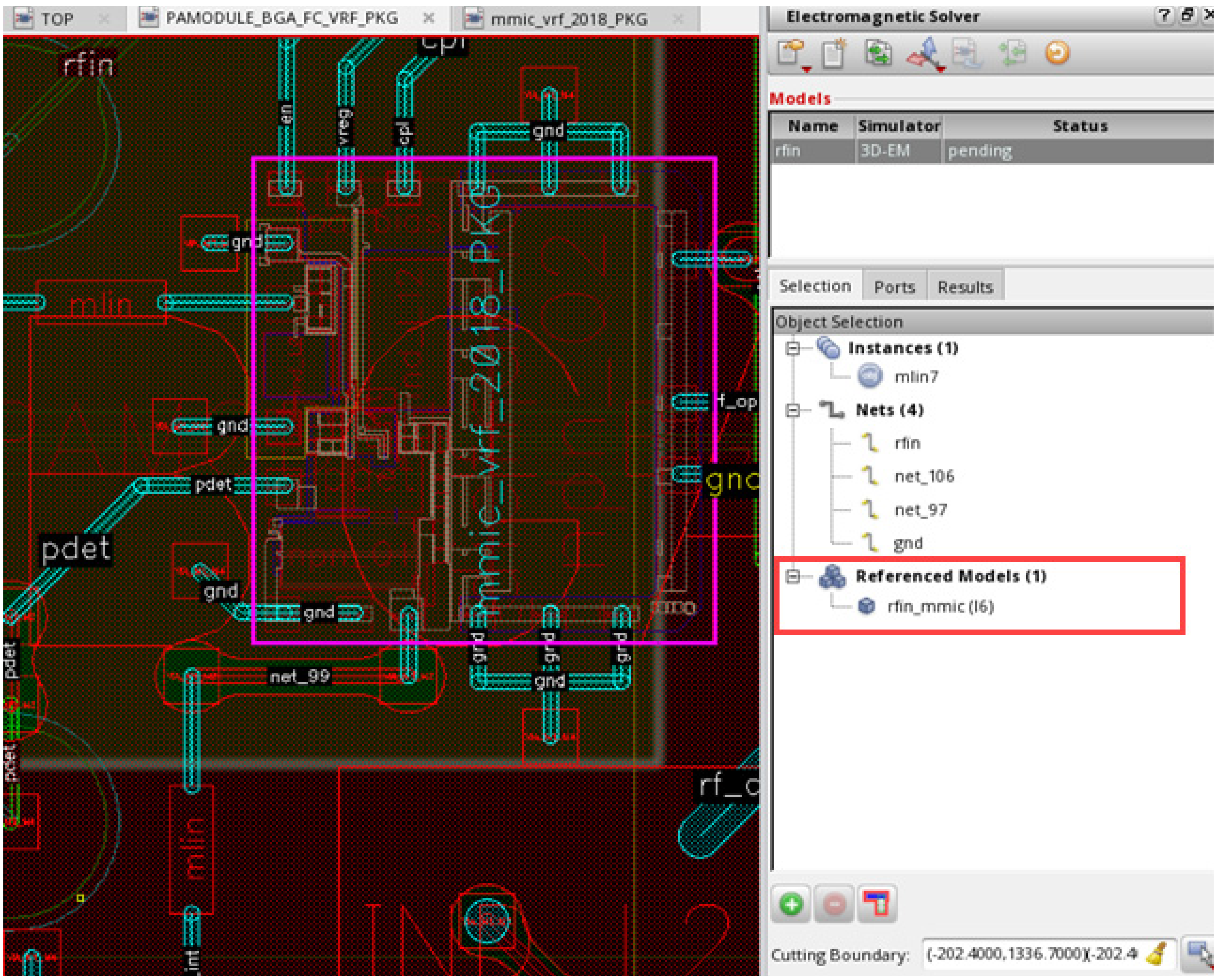
If the list of EM models is empty, ensure that the models you want to reference are using the Clarity 3D simulator and have the *Current Return Path* set as `Local Reference`.

d. Select a model to be included for the die abstract.

e. Click *OK*.

© 2023 Cadence Design Systems, Inc.

The selected model is shown in the *Referenced Models* tree on the *Selection* tab of the Electromagnetic Solver assistant, as shown below.



9. Similarly, create models for all fabrics in the design and add reference for the die abstracts to be included in the model.

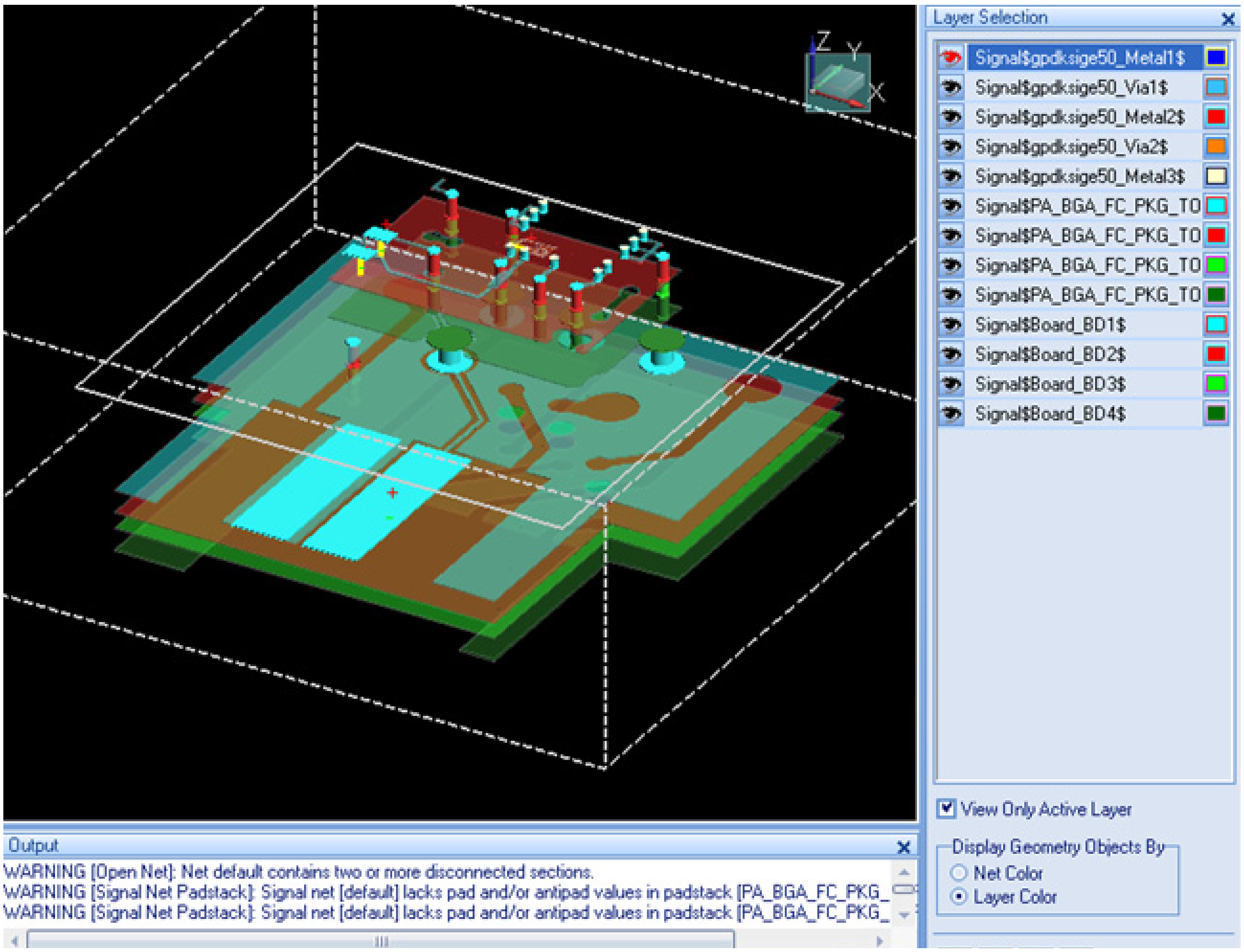
10. After defining the model at the top level, choose *Simulation – Create and Edit* to extract a `.clf` file and send it to Clarity 3D Solver.

    During this step, Virtuoso processes the selected shapes according to the settings given in the models. For example, it merges the IC stack, which appears as one in the Layer Manager in Clarity 3D Solver.

11. Enable the 3D view in Clarity 3D window, and validate the fabric layers and selected objects.

© 2023 Cadence Design Systems, Inc.

For the above example, the 3D view of the model appears as shown below.



12. Run simulation in Clarity 3D Layout to create n-ports that are saved in an S-param touchstone file.

The touchstone file produced by cross-fabric EM simulation contains a header with all nets and instances from all fabrics.

13. Review the resulting ports on the *Results* tab of the Electromagnetic Solver assistant and proceed with the creation of an extracted view as explained in Creating Extracted Views from Models that you can use to run simulations in Virtuoso ADE Assembler.

© 2023 Cadence Design Systems, Inc.

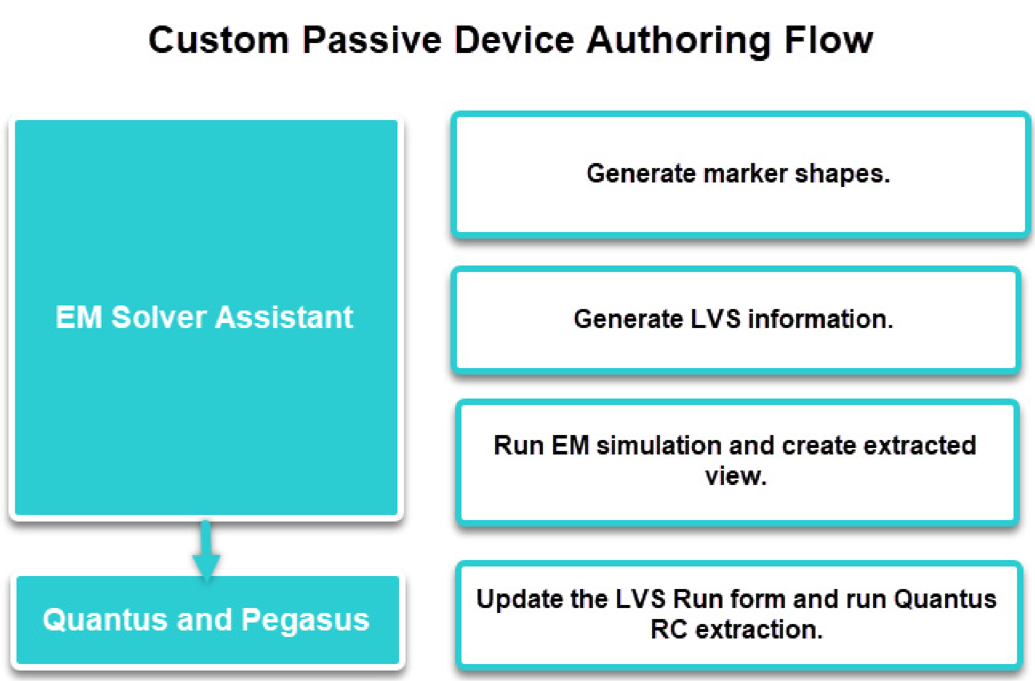
© 2023 Cadence Design Systems, Inc.

5

Authoring Custom Passive Devices

To optimize custom devices (passive devices) that are not part of the foundry process, RFIC designers need to create these devices manually in the layout. Designers can use EMX in full cellview mode to characterize the device, and create symbol and `sparam` views.

When the device is used in a design, the symbol is placed in the schematic and the `sparam` view can be selected in the Hierarchy Editor for circuit simulation. The newly created device is also required to support layout generation and physical sign-off.

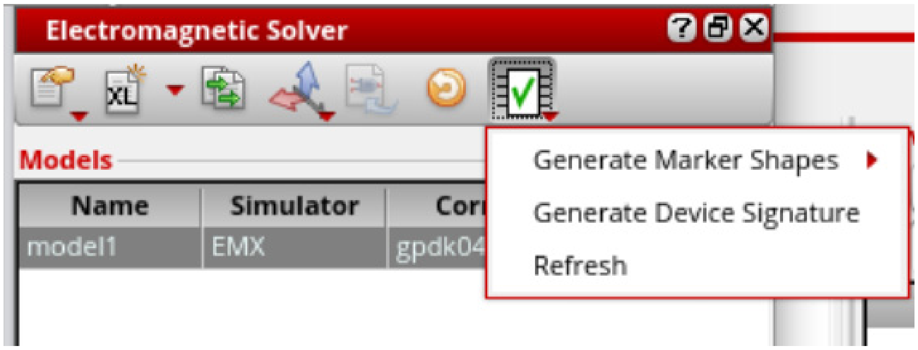


To enabling custom passive device for LVS:

1. Create the devices manually in the layout.
2. Open the layout in Layout MXL and set the workspace to `Electromagnetic`.

© 2023 Cadence Design Systems, Inc.

3. Click *Device Authoring – Generate Marker Shapes* from the toolbar of the Electromagnetic Solver assistant to generate marker shape over the created device by using the `device.txt` file.



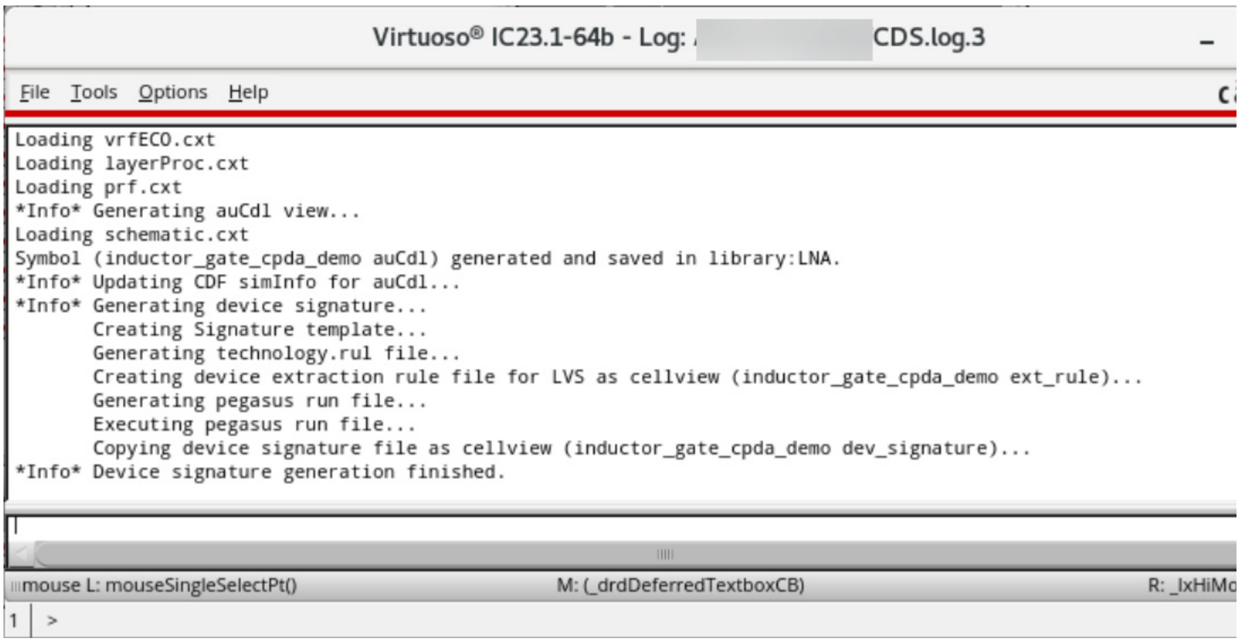
A marker shape is required to cut device metal from an interconnect metal to avoid shorts and double-counting of parasitics. Such shapes can be further used for design rule checks (DRC)/Metal Fills process.

DRC are run by the DRC tool to verify whether a layout conforms to the minimum width, spacing, and other design rules established for a given fabrication process or process variation. Metal Fill allows you to generate dummy fills using foundry fill decks and automatically map generated dummy fills into the Virtuoso OA database.

4. Click *Device Authoring – Generate Device Signature* by using the `DeviceMap.txt` file, which contains information required to generate device

© 2023 Cadence Design Systems, Inc.

signature, such as recognition layer, marker layer information, pin mapping information, and context layers used in the design.



5. Generate ports.
6. Run simulation and load the results.
7. Create an extracted view.

Related Topics

cpdCreateDeviceInfo

cpdGenerateDerivedShapes

cpdLoadDerivedShapesRules

cpdPegasusPreFormTrigger

Variables for the Custom Passive Device Authoring Flow

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

A

The Electromagnetic Solver Assistant GUI

The Electromagnetic Solver assistant user interface has the following components:

Field	Description
Electromagnetic Solver Assistant Toolbar	Lets you access the important commands required to run electromagnetic simulations.
Models Table	Lets you create or manage EM models for the current layout cellview.
Selection Tab	Lets you specify objects and boundaries to be used for the model selected in the *Models* table.
Setup Tab (IC Layout Only)	Lets you create or manage EM models for the current layout cellview.
Ports Tab (EMX)	Lets you specify ports and edges for a model created for an IC layout.
Ports Tab (Clarity with local reference)	Lets you specify ports and edges for a model created for package layouts using the Clarity 3D Solver with local reference.
Ports Tab (Clarity with PEC Ground Ring)	Lets you specify ports for a model created for package layouts using the Clarity 3D Solver with PEC Ground Ring.
Pre-Process Tab	Lets you specify options to prepare or process the layout before running electromagnetic simulation. You can also review the geometries in the model to be sent to the EM solver
Results Tab	Shows the path to the results saved after an EM simulation is run for a model. It also provides commands to plot the results.

© 2023 Cadence Design Systems, Inc.

Electromagnetic Solver Assistant Toolbar

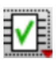
The following table describes the Electromagnetic Solver assistant toolbar buttons.

Icon	Command	Description
	*Settings*	Lets you configure settings for simulator or environment settings. It also provides commands that let you view stackup for models.
	Simulation	Opens the Simulation Settings form where you can configure settings for different 3D electromagnetic simulators.
	Environment	Opens the Environment Settings form that lets you review and update the process corner name and custom process corner files for your models. You can also specify the settings to be used by the jobs that run simulations.
	View Stackup	Opens the Virtuoso Layout Viewer window and displays the layer stackup as defined in the `.emproc` files for the models that use the Clarity simulator. **Note:** This command is visible for only when you select a model that uses the Clarity simulator.
	View EMX Scaled Stackup	Opens the PDF viewer and displays the scaled layer stackup for the models that use the EMX simulator. **Note:** This command is visible for only when you select a model that uses the EMX simulator.
	View EMX Unscaled Stackup	Opens the PDF viewer and displays the unscaled layer stackup for the models that use the EMX simulator. **Note:** This command is visible for only when you select a model that uses the EMX simulator.
	*New Full Cellview Model*	Creates a new model based on full cellview extraction. A new row is also added to the *Models* table.

© 2023 Cadence Design Systems, Inc.

Icon	Command	Description
	New Nets/Instances Model (XL)	Creates a new model on XL-based selection. A new row is also added to the *Models* table.
	New Nets/Instances Model (LVS)	Creates a new model on LVS-based selection. A new row is also added to the *Models* table.
	*Export model settings for use in the EM solver*	Exports the model settings to a `.clf` file that can be used by the EM solver specified for that model in the *Models* table.
	*Open simulator*	Provides commands to interact with the electromagnetic solvers.
	Preview Geometry	Provides the preview of a model in EMX gdsview. It exports a CLF file and displays it in gdsview and shows the layout exactly as the EMX solver sees it.
	Open Existing Model	Reopen the result of a previous simulation in Clarity. **Note:** This command is visible for only when you select a model that uses the Clarity simulator.
	Generate Mesh	Creates a mesh file for the selected model. **Note:** This command is visible for only when you select a model that uses the EMX simulator.
	View 3D Mesh	Opens a mesh in a mesh viewer. **Note:** This command is visible for only when you select a model that uses the EMX simulator.
	Mesh and Simulate	Creates a mesh for the selected model, sends the mesh file to the selected simulator, opens the simulator UI and runs simulation. **Note:** This command is visible for only when you select a model that uses the EMX simulator.
	Start Simulation	Starts the simulation. **Note:** This command is visible for only when you select a model that uses the Clarity simulator.
	Stop	Stops the simulation in progress.

© 2023 Cadence Design Systems, Inc.

Icon	Command	Description
	View Simulation Log	Opens the log saved by the simulator. **Note:** This command is visible for only when you select a model that uses the EMX simulator.
	*Create Extracted View*	Creates/ an extracted view by using the S-parameter saved after the EM simulation of a model.
	*Refresh Model Status*	Reflects the current status of model simulation. If the EM simulation is complete, but the value in the Status column is still shown as `Pending`, use this command to update the status,
	*Device Authoring*	Provides commands to enable custom passive devices for the signoff LVS. This option is turned on only if the *Extract Full Cellview* check box is selected.
	Generate Marker Shapes	Generates marker shapes over the created device using logical operations.
	Generate Device Signature	Generates device signature, such as recognition layer, marker layer information, pin mapping information, and layers used in the design.
	Refresh	Reloads the updated `devices.txt` file during a Virtuoso session.

Related Topics

Viewing the Layer Stackup for a Model

Configuring Settings for Electromagnetic Simulations

Configuring Process Settings for IC Layouts

Viewing the Layer Stackup for a Model

© 2023 Cadence Design Systems, Inc.

Models Table

The following table describes the columns available in the *Models* table of the Electromagnetic Solver assistant.

Column	Description
Name	Name of the model.
Simulator	Simulator selected to run EM simulations for the model.
Corner	Name of the process corner.
Status	Status of the model. Possible values are: ■ `pending`: When you are configuring a model and electromagnetic simulation is not run. ■ `running`: When the electromagnetic simulation is in progress. ■ `done`: After the simulation is complete. ■ `exported`: After you export the S-parameter file to a schematic or S-param view.

Related Topics

Creating a Model

© 2023 Cadence Design Systems, Inc.

Selection Tab

The following table describes the columns available in the *Selection* tab of the Electromagnetic Solver assistant.

Field/Command		Description
Object Selection		Displays the instances, nets, and reference models that you select for inclusion in a model. **Note:** This list is not available when you select the *Extract Full Cellview* check box.
Add selected instances, nets, or reference models		Enables extraction of a full cellview for a model.
Remove instances, nets, or reference models		Enables extraction of a full cellview for a model.
Include devices between nets		Includes any schematic instance, such as LVS resistors and net splitters, where all pins are connected to nets in the model automatically. **Note:** This command is only for LVS models.
SVDB Directory		Specifies a directory to load the SVDB data into memory. **Note:** This command is only for LVS models.
Excluded Cells		Specifies a list of cells to be excluded when you are using a full cellview for a model. **Note:** This list is available only when you select the *Extract Full Cellview* check box.
Edit the list of excluded cells		Lets you edit the *Excluded Cells* list. **Note:** This list is available only when you select the *Extract Full Cellview* check box.
Highlight shapes inside the cutting boundary except the excluded cells		Highlights the selected shapes when you are using a full cellview for a model.

© 2023 Cadence Design Systems, Inc.

Field/Command	Description
Cutting Boundary	Shows the coordinates of the cutting boundary that you choose for a model. **Note:** This command is not available for LVS models.
Select cut type 	Lets you specify a shape on the layout canvas. **Note:** This command is not available for LVS models.
Draw Rectangle	Changes the mouse pointer and lets you select a rectangular boundary.
Draw Polygon	Changes the mouse pointer and lets you select a rectangular boundary.
Show Cutting Boundary 	Highlights the cutting boundary in the layout canvas. **Note:** This command is not available for LVS models.
Config File	Specifies the path to the config file that contains the detail of the material file to be used for a package layout. **Note:** This field is visible for only a package layout.

Related Topics

Selecting Objects from a Package Layout

Selecting Objects from an IC Layout

Highlighting Selected Objects on the Layout Canvas

© 2023 Cadence Design Systems, Inc.

Setup Tab (IC Layout Only)

The following table describes the fields available in the *Setup* tab of the Electromagnetic Solver assistant.

Note: This tab is visible for only when you use the Clarity simulator.

Field	Description
Die Ground	Specifies the scale type used while defining the size of a metal shape that is created at the bottom of the substrate.
	Specifies the scale percentage for the die ground. This scale factor is applied around the bounding box of the model, which is the bounding box of all selected nets and instances, optionally trimmed by a cutting boundary.
Display Highlight	Displays or hides a flashing purple square on the layout canvas to represent the X and Y extents of the box to be used as the simulation region
Side Walls	Specifies the type of side walls of the die ground.

Related Topics

Specifying Die Ground Settings for an IC Layout

© 2023 Cadence Design Systems, Inc.

Ports Tab (EMX)

Important

For an IC layout, the *Ports* tab shows the port and edge details for a model.

The following table describes the details available in the *Ports* tab for an IC layout cellview using the EMX simulator.

Table/Command	Description
Ports Table	This table shows the details of ports to be included in a model.
- *Reference*	Names of the reference nets to be used for the ports. **Note:** This field is available only for Clarity models when the *Current Return Path* field on the *Selection* tab is set to *Local Reference*.
- *Instance*	Name of the instance for which a port is created.
- *Pin*	Name of a instance pin to which a port is connected.

© 2023 Cadence Design Systems, Inc.

Table/Command	Description
- Type	Possible values for a model that uses the Clarity simulator: ■ `Cross layer`: The positive and negative edges for these ports are placed on different layers. These ports connect the layers vertically. ■ `Same layer`: The positive and negative edges for these ports are placed horizontally on the same layer. ■ `Diagonal`: The positive and negative edges for these ports are placed diagonally on different layers. Possible values for a model that uses the EMX simulator: ■ `Included`: (Default) The port is to be included in the simulation. You can make it a differential port by specifying a reference pin in the *Reference* column. ■ `Grounded`: There is no port. The specified pin is directly connected to ground. When you add grounded pins to the layout, the model has fewer ports and the simulation runs faster. ■ `Disabled`: The port is disabled and is not used in simulation, but it is used as a reference for other ports of type `Included`. To use a disabled port as a reference for another port, specify it in the *Reference* column for that port.
- Net	Name of the net to which the port is connected.
Automatically generate ports	Automatically generate ports for the instances selected on the *Selection* tab. The rules for port generation vary for IC and package layouts and also for different EM simulators.
Add Port	Adds a new row in the ports table.
Remove Port	Removes the selected port from the ports table.
Move Port Up	Move the selected ports up in the ports table.
Move Port Down	Move the selected ports down in the ports table.

© 2023 Cadence Design Systems, Inc.

Table/Command	Description
Report Schematic Binding	Displays the schematic binding report. By referring to this report, you can resolve any XL binding issues before running a simulation. **Note:** This command is not visible when *Extract Full Cellview* is selected on the *Selection* tab.
Validate Ports	Runs the rules that validate the ports.
N-Port Common Node	Specifies the name of a net to which the reference pin of the n-port instance in the schematic is connected
Edges Table	This table shows the details of edges of a port selected in the ports table. **Note:** This table is available only for IC layouts when the model is using EMX.
- Polarity	Specify + or - that indicates the polarity of an edge.
- T	Specifies the connection type for an edge.
- Layer	Name of the layer to which an edge of a port is connected.
- X1	X coordinate of the top-left point of an edge.
- Y1	Y coordinate of the bottom-right point of an edge.
- X2‘	X coordinate of the top-left point of an edge.
- Y2	Y coordinate of the bottom-right point of an edge.
Move Connection	Lets you move the selected edge.
Add Edge Connection	Lets you select an edge at which you want to create an edge connection. It adds a new row in the edges table and fills in the details of the edge.
Add Point Connection	Lets you select a pin for which you want to create an internal port. The tool creates a port at the pin location. The size of the port is same as that of the pin.
Remove Connection	Removes the selected row from the edges table.

Related Topics

Specifying Ports for EMX Models in IC Layouts

© 2023 Cadence Design Systems, Inc.

Ports Tab (Clarity with local reference)

For package layouts using the Clarity 3D Solver with local reference, the *Ports* tab contains two sub-tabs:

- *Setup*: Contains the ports table
- *Components*: Shows the components or instances connected to the nets selected for a model. This tab is visible only when Clarity 3D Solver is used for a model.

Setup Sub-Tab

The following table describes the fields available in the *Setup* sub tab of the *Ports* tab.

Table/Command	Description
Reference Nets	The nets in this list are treated as ground nets for the current return. This field is used for automatic port generation.
Ports Table	This table shows the details of ports to be included in a model.
- Instance	Name of the instance for which a port is created.
- Pin	Name of a instance pin to which a port is connected.

© 2023 Cadence Design Systems, Inc.

<table>
<tr><th>Table/Command</th><th>Description</th></tr>
<tr><td>- Type</td><td>Possible values for a model that uses the Clarity simulator:
<code>Cross layer</code>: The positive and negative edges for these ports are placed on different layers. These ports connect the layers vertically.
<code>Same layer</code>: The positive and negative edges for these ports are placed horizontally on the same layer.
<code>Diagonal</code>: The positive and negative edges for these ports are placed diagonally on different layers.
Possible values for a model that uses the Clarity simulator:
<code>Included</code>: (Default) The port is to be included in the simulation. You can make it a differential port by specifying a reference pin in the Reference column.
<code>Grounded</code>: There is no port. The specified pin is directly connected to ground. When you add grounded pins to the layout, the model has fewer ports and the simulation runs faster.
<code>Disabled</code>: The port is disabled and is not used in simulation, but it is used as a reference for other ports of type <code>Included</code>. To use a disabled port as a reference for another port, specify it in the Reference column for that port.</td></tr>
<tr><td>- Width</td><td>The width of the port.</td></tr>
<tr><td>- Net</td><td>Name of the net to which the port is connected.</td></tr>
<tr><td>Automatically generate ports</td><td>Automatically generate ports for the instances selected on the Selection tab. The rules for port generation vary for IC and package layouts and also for different EM simulators.</td></tr>
<tr><td>Add Port</td><td>Adds a new row in the ports table.</td></tr>
<tr><td>Remove Port</td><td>Removes the selected port from the ports table.</td></tr>
<tr><td>Move Port Up</td><td>Move the selected ports up in the ports table.</td></tr>
<tr><td>Move Port Down</td><td>Move the selected ports down in the ports table.</td></tr>
</table>

© 2023 Cadence Design Systems, Inc.

Table/Command	Description
Report Schematic Binding	Lets you check the schematic binding and resolve any XL binding issues before running a simulation. **Note:** This command is only for XL models.
Validate Ports	Runs the rules that validate the ports.
N-Port Common Node	Specifies the name of a net to which the reference pin of the n-port instance in the schematic is connected
Edges Table	**Note:** This table shows the details of edges of a port selected in the ports table.
- *Layer*	Name of the layer to which an edge of a port is connected.
- *X*	The X coordinate of the location of the port connection.
- *Y*	The Y coordinate of the location of the port connection.
Rotation	The rotation value of the cross-fabric ports.

© 2023 Cadence Design Systems, Inc.

Components Sub-Tab

The following table describes the instance types and their properties you can set for a model in the *Components* sub-tab of the *Ports* tab.

Instance Type	**Port Properties**
*I*C or Package	*Type*: You can choose from the following port types: ■ *None*: Does not create any port for the instance ■ *Coax*: Creates a coaxial port between each instance pin and the reference plane. The bumps and balls required for this coaxial connection are also created. ■ *Vertical*: Creates a vertical port between each instance pin and the reference plane. No balls or bumps are required for this connection. ■ *Vertical + bumps/balls*: Creates a bump or a ball on the instance pin. Then, it creates a vertical port between the bump or ball and the reference plane. Default value: *Coax* **Note:** The ports for IC or package instances are created by Clarity 3D Solver while exporting the model. These are not visible in Virtuoso and are not considered for the port validation run by the Electromagnetic Solver assistant. *Port size increase*: Specifies the percentage value by which the tool should increase the port size. This is applicable for coaxial ports only. For example, if the port size increase is set to 50%, and the bump diameter is 100um, then the diameter of the circular cut-out around the bump would be 1.4 times the bump diameter, or 140um. *Port Height*: Specifies the height of the vertical ports. This is applicable for vertical ports only. *Bump Parameters*: Opens the Ball and Bump Editor form where you can specify the parameter values that the tool can use to create balls or bumps for the *Coax* or *Vertical + bumps/ balls* port types.

© 2023 Cadence Design Systems, Inc.

Instance Type	Port Properties
SMD	*Type*: You can choose from the following port types: ■ *None*: Does not create any port for the instance ■ *Coax*: Creates a coaxial port between each instance pin and the reference plane. The bumps and balls required for this coaxial connection are also created. ■ *Vertical*: Creates a vertical port between each instance pin and the reference plane. No balls or bumps are required for this connection. ■ *Vertical + bumps/balls*: Creates a bump or a ball on the instance pin. Then, it creates a vertical port between the bump or ball and the reference plane. ■ *Same Layer/Cross Layer*: The tool searches the layers above and below the SMD pad for a ground reference. If found, it creates a cross-layer port. If not, the tool searches for a ground reference on the same layer. If a reference is found, it creates a same-layer port. **Note:** The ports for SMD instances are created by Clarity 3D Solver while exporting the model. These are not visible in Virtuoso and are not considered for the port validation run by the Electromagnetic Solver assistant.
Other	*Type*: You can choose from the following port types: ■ *None*: Does not create any port for the instance ■ *Same Layer/Cross Layer*: The tool searches the layers above and below each pin for a ground reference. If found, it creates a cross-layer port. If not, the tool searches for a ground reference on the same layer. If a reference is found, it creates a same-layer port.

Related Topics

Specifying Ports for Clarity Models in Package Layout

© 2023 Cadence Design Systems, Inc.

Ports Tab (Clarity with PEC Ground Ring)

The following table describes the details available in the *Ports* tab for package layouts using the Clarity 3D Solver with PEC Ground Ring.

Setup Sub-Tab

The following table describes the fields available in the *Setup* sub tab of the *Ports* tab.

Table/Command	Description
Ports Table	This table shows the details of ports to be included in a model.
- Instance	Name of the instance for which a port is created.
- Pin	Name of a instance pin to which a port is connected.
- Layer	Name of the layer to which a port is connected.
- Net	Name of the net to which the port is connected.
Automatically generate ports	Automatically generate ports for the instances selected on the *Selection* tab. The rules for port generation vary for IC and package layouts and also for different EM simulators.
Add Port	Adds a new row in the ports table.
Remove Port	Removes the selected port from the ports table.
Move Port Up	Move the selected ports up in the ports table.
Move Port Down	Move the selected ports down in the ports table.
Report Schematic Binding	Lets you check the schematic binding and resolve any XL binding issues before running a simulation. **Note:** This command is only for XL models.
Validate Ports	Runs the rules that validate the ports.

Related Topics

Specifying Ports for Clarity Models in Package Layout

© 2023 Cadence Design Systems, Inc.

Pre-Process Tab

The following table describes the controls available in the *Pre-Process* tab of the Electromagnetic Solver assistant.

Field/Command	Description
Via Cluster	This section provides options to cluster vias. **Note:** This group is applicable for only Clarity models.
Cluster Vias	Enables via clustering for your model.
Configure	Opens the Via Clustering Settings - *<model-name>* form.
Merge non-Manhattan Vias	Lets you ignore the *Max Cuts* via cluster setting so that you can cluster the maximum possible vias.
Setup	This section provides options to specify settings for the simplification of certain shapes on the layout that are not expected to affect the results of electromagnetic extraction. This improves the performance of the extraction run because the number of unknown shapes to be solved is reduced.
Setup	Opens the Simplify Layout for EM Simulation form in which you can specify the rules for shape simplification.
Select file	Specifies the path to the recipe file in which you have saved the shape simplification setup.
Protect	This section provides options to protect certain shapes from getting modified by the pre-processing tasks specified on this tab. You can choose shapes that are relevant RF signals, vias, and instances and must be considered for electromagnetic simulation.
Protect Selected	Protects the selected shapes from shape simplification.
Unprotect Selected	Removes shape simplification protection from the selected shapes.
Highlight Protected	Highlights on the layout canvas the shapes that are protected from shape simplification.
Preview	This section provides the command to preview the EM layout that will be sent to the solver for simulation

© 2023 Cadence Design Systems, Inc.

Field/Command	Description
Preview EM Layout	Displays the preview of the selected model in the EM Preview of Model <model-name> window.

Related Topics

Specifying Options for Via Clustering

Specifying Shape Simplification Options for IC Layouts

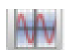
Protecting Layout Objects from Shape Simplification

Previewing Pre-Processed Layout

© 2023 Cadence Design Systems, Inc.

Results Tab

The following table describes the fields available on the *Results* tab of the Electromagnetic Solver assistant.

Column		Description
Run Status		Displays the current status of the electromagnetic simulation for the model selected in the *Models* table.
S-Params File		Displays the path where Layout MXL saved the S-parameter file.
Open in Results Browser		Opens the data from S-Parameters file into Results Browser of Virtuoso® Visualization and Analysis XL.
Open in text viewer		Opens the data from S-Parameters file into a text viewer
Copy path to clipboard		Opens the data from S-Parameters file into

Related Topics

Validating S-Parameter Data Saved by an EM Simulation

© 2023 Cadence Design Systems, Inc.

B

Electromagnetic Solver Assistant Forms

This section lists the forms related to the Electromagnetic Solver assistant.

- Backannotate from Extracted View Form
- Create S-Parameter View Form
- Create Simulation Schematic Form
- Environment Settings Form
- Load Simulation Settings Form
- Save Simulation Settings Form
- Set Port Width Form
- Set Reference Pin Form
- Simplify Layout for EM Simulation Form
- Simulation Settings Form
- Via Clustering Settings Form

© 2023 Cadence Design Systems, Inc.

Backannotate from Extracted View Form

Use the Backannotate from Extracted View form to highlight the model in the schematic where you can review the S-parameter instances and nets.

Field	Description
Extracted View	Specifies the name of the extracted view from which you want to highlight parameters in the schematic.
Table of models	Displays a list of N-port instances and the models available in the selected extracted view.

Related Topics

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Create S-Parameter View Form

Use the Create S-Parameter View form to save a model as an S-parameter cellview.

Field	Description
Library	Specifies the name of the library in which you want to save the extracted view. Related environment variable: exportLibName
Cell	Specifies the name of the cell in which you want to save the extracted view. Related environment variable: exportCellName
View	Specifies the name of the new S-parameter view to be saved. Related environment variable: exportSparamViewName
Create Symbol	Specifies whether to save a symbol view in addition to the S-parameter view in the same library and cell.

Related Topics

Creating S-Parameter Cellviews for Models

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Create Simulation Schematic Form

Use the Create Simulation Schematic form to create a schematic using the S-Parameters.

Field	Description
Library	Specifies the name of the library in which you want to save the new schematic view.
Cell	Specifies the name of the cell in which you want to save the new schematic view.
View	Specifies the name of the new schematic view to be saved.
Create Symbol	Specifies whether to save a symbol view in addition to the schematic view in the same library and cell.
Update Existing Symbol	Specifies whether to update the existing symbol view, if available.

Related Topics

Creating an S-Parameter Model for a Layout-Driven Flow

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Environment Settings Form

Use the Environment Settings form to review and update the process corner name and custom process corner files for your models. You can also specify the settings to be used by the jobs that run simulations.

Field	Description
EMX	This section provides options to specify process corner settings for the models that use EXM Planar 3D solver.
Process Corner Directory (.proc files)*	Specifies the path to the directory where `.proc` files with process corner settings are saved. By default, the Electromagnetic Solver assistant looks for the files in `.cadence/dfII/Sigrity/corners`. You can specify an absolute path or a relative path to the process corner directory. For relative paths, the current working directory of Virtuoso is considered as the base directory. Related environment variable: emxProcessCornerDirectory
Default Process Corner	Specifies the name of the default corner to be used for a new model that uses the EMX solver. This drop-down list displays the names of all `.proc` files available in the process corner directory. You can choose a file from the given list. Related environment variable: emxProcessDefaultCorner
EMX GDS Layer Map	Specifies the path to the GDS layer map file to be used when sending models to the EMX solver. By default, the tool uses the GDS layer map provided in the technology library. Related environment variable: emxLayerMap
Clarity	This section provides options to specify process corner settings for the models that use Clarity 3D Solver.

© 2023 Cadence Design Systems, Inc.

Field	Description
Process Corner Directory (.emproc files)*	Specifies the path to the directory where `.emproc` files with process corner settings are saved. By default, the Electromagnetic Solver assistant looks for the files in `.cadence/dfII/Sigrity/corners`. You can specify an absolute path or a relative path to the process corner directory. For relative paths, the current working directory of Virtuoso is considered as the base directory. Related environment variable: processCornerDirectory
Default Process Corner	Specifies the name of the default corner to be used for a new model that uses the Clarity 3D Solver simulator. This drop-down list displays the names of all `.emproc` files available in the process corner directory. You can choose a file from the given list. Related environment variable: processDefaultCorner
Jobs	This section contains the settings to be used by the jobs that run simulations.
Run directory	Specifies the run directory for all the models created in the Electromagnetic Solver assistant. The tool also saves the `.clf` file for models at this location. You can use shell environment variables in the path specified by this variable. By default, this field is set to *`Virtuoso-current-working-directory`*`/.cadence/dfII/Sigrity`. Related environment variable: runDirectory
Remote job submission command	Specifies the DRMS (Distributed Resource Management System) command to be used by the EM solvers to distribute jobs to a workload job scheduling software, such as LSF or OpenLava Related environment variable: drmsCommand
File	Specifies the path of the file in which you want to save the environment settings specified in this form. Later, you can load the saved settings in the same or a different cellview.
Browse	Browses to select a file.

© 2023 Cadence Design Systems, Inc.

Field	Description
Save To	Saves the environment settings in the file specified in the *File* field.
Load From	Loads the environment settings from the file specified in the *File* field.
LVS	This section contains the settings to be used by the LVS flow to specify the layer map files.
LVS Layer Map	Specifies the location of the LVS layer map file. This file is required when using LVS-based models because it contains the mapping from LVS layers to Virtuoso layers. Environment variable: lvsLayerMapFile
Quantus Layer Setup File	Specifies the path to the `layer_setup` file in the Quantus enablement. This file is part of the PDK. You do not need to create it. It is needed only when extracting partial nets. Environment variable: quantusLayerSetupFile
Quantus TRP File	Specifies the path to the Quantus TRP file. Not all PDKs have a TRP file. It is needed only when device pin names are different on LVS layers and Virtuoso layers. Environment variable: quantusTrpFile

Related Topics

Configuring Process Settings for IC Layouts

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Load Simulation Settings Form

Use the Load Simulation Settings Form to load the simulation settings saved in a file.

Field	Description
File Name	Specifies the name of the file from which you need to load the simulation settings.
Files of type	Specifies the file type to display the files of a specific type.

Related Topics

Configuring Settings for Electromagnetic Simulations

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Save Simulation Settings Form

Use the Save Simulation Settings Form to save the simulation settings saved in a file.

Field	Description
File Name	Specifies the name of the file from which you need to load the simulation settings.
Files of type	Specifies the file type to display the files of a specific type.

Related Topics

Configuring Settings for Electromagnetic Simulations

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Select EM Models Form

Use the Select EM Models form to choose a reference model for an instance selected for another model in a cross-fabric design..

Field	Description
EM Models for Selected Instance	Displays a list of all the models available for the instance selected on the Selection tab of the Electromagnetic Solver assistant.
OK	Successfully completes the selection of a reference model.
Cancel	Closes the form without selection of a reference model.

Related Topics

Extracting Models for a Cross-Fabric Design

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Set Port Width Form

Use the Set Port Width form to specify the width of the ports selected on the *Ports* tab for a model that uses Clarity 3D Solver.

Field	Description
Port Width	Specifies a value greater than 0.
OK	Successfully accepts the specified port width.
Cancel	Closes the form without using the specified port width.

Related Topics

Creating Ports Manually for the Models that use Clarity 3D Solver

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Set Reference Pin Form

Use the Set Reference Pin form to specify a reference pin for the ports selected on the *Ports* tab for a model that uses the EMX simulator.

Field	Description
Reference Pin	Specifies a name of a reference pin.
OK	Successfully accepts the specified reference pin.
Cancel	Closes the form without using the specified reference pin.

Related Topics

Specifying Ports for EMX Models in IC Layouts

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Simplify Layout for EM Simulation Form

Use the Simplify Layout for EM Simulation form to specify the rules for shape simplification.

Field	Description
Skip all shapes on LPP	This section provides options to create shape simplification rules based on their fill size to skip dummy metal fills from one or more layer-purpose pairs.
List of rules	Displays all the rules created in this section
Lpp	Specifies the layer-purpose pairs for which you want to create a rule. You can choose from the drop-down list that displays all the layer-purpose pairs from the design.
Fill size	Specifies the metal fill size to be used for the rule
Add	Adds a new rule to the list
Delete Selected	Deletes the selected rules from the list
Clear	Removes all the rules from the list
Remaster Instances	This section provides options to create rules to change the master for instances in the layout. For example, you may want to run a simulation by using a manually simplified version of a cellview instead of the actual cellview used in the design.
List of rules	Displays all the rules created in this section
From	Contains three drop-down lists to specify the names of the current library, cell, and view master of the instances
To	Contains three drop-down lists to specify the names of the current library, cell, and view to which the instances belong
Add	Adds a new rule to the list
Delete Selected	Deletes the selected rules from the list
Clear	Removes all the rules from the list
Remove dangling shapes	This section provides options to create rules based on the layer size to remove dangling connected shapes from one or more layer-purpose pairs in the layout
List of rules	Displays all the rules created in this section

© 2023 Cadence Design Systems, Inc.

Field	Description
Lpp	Specifies the layer-purpose pairs for which you want to create a rule. You can choose from the drop-down list that displays all the layer-purpose pairs from the design.
Min Layer Size	Specifies the minimum size of shapes you want to retain on the layer. The tool will remove any shape smaller than the specified size.
Direction	Specifies the direction in which the minimum size of shapes is measured.
Add	Adds a new rule to the list
Delete Selected	Deletes the selected rules from the list
Clear	Removes all the rules from the list
Merge shapes	This section provides options to create rules to merge shapes that are placed on the same layer or on the layers above or below it if they are within the given distance.
List of rules	Displays all the rules created in this section
Lpp	Specifies the layer-purpose pairs for which you want to create a rule. You can choose from the drop-down list that displays all the layer-purpose pairs from the design.
Spacing	Specifies the minimum allowed distance between the devices
Add	Adds a new rule to the list
Delete Selected	Deletes the selected rules from the list
Clear	Removes all the rules from the list
Stripe shapes	This section provides options to create rules to convert layers into stripes of the given width. This helps in converting fine mesh into a coarse mesh that would contain less number of unknowns to be passed to the solver.
List of rules	Displays all the rules created in this section
Spacing	Specifies the spacing to kept between each stripe
Width	Specifies the width for stripes
Lpp	Specifies the layer-purpose pairs for which you want to create a rule. You can choose from the drop-down list that displays all the layer-purpose pairs from the design.

© 2023 Cadence Design Systems, Inc.

Field	Description
Add	Adds a new rule to the list
Delete Selected	Deletes the selected rules from the list
Clear	Removes all the rules from the list
Smoothing shape steps	This section provides options to create rules to smoothen out small steps or edges after the layers are striped. This also helps in reducing the number of unknowns in the layout.
List of rules	Displays all the rules created in this section
Lpp	Specifies the layer-purpose pairs for which you want to create a rule. You can choose from the drop-down list that displays all the layer-purpose pairs from the design.
Step size	Specifies the minimum allowed step size. All the steps with size less than the given limit are smoothed.
Add	Adds a new rule to the list
Delete Selected	Deletes the selected rules from the list
Clear	Removes all the rules from the list

Related Topics

Specifying Shape Simplification Options for IC Layouts

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Simulation Settings Form

The Simulation Settings form lets you specify the settings for the various simulators you can use with the Electromagnetic Solver assistant. The form contains the following tabs.

Tab	Description
The General Tab in the Simulation Settings Form	Lets you specify the common settings used by all simulators.
The Clarity Tab in the Simulation Settings Form	Lets you specify the common settings used by Clarity 3D Solver.
The EMX Tab in the Simulation Settings Form	Lets you specify the common settings used by EMX Planar 3D Solver.
The LVS Tab in the Simulation Settings Form	Lets you specify the settings used by the LVS flow.

Note: The EMX simulator is supported only for IC layouts. Therefore, the tabs for these simulators are available on this form only when you open an IC layout in Layout MXL.

The General Tab in the Simulation Settings Form

The following table describes the general settings that can be used by all simulators.

Field	Description
Frequency Sampling	This section provides options to specify frequency settings to be used by the simulator
Advanced Frequency Sweep	Enables fast, but accurate computation. When cleared, the simulator uses point-by-point frequency sweep and takes longer to complete the simulation. Default value: `Selected`
Explicit DC Solution	Enables calculation of S-Parameters at DC. The DC data is saved in the Touchstone file format. Sets the minimum frequency to `0` and calculates the S-parameter at DC.

© 2023 Cadence Design Systems, Inc.

Field	Description
Frequency Min (Hz)	Sets the minimum frequency for the frequency sweep. For DC, set this field to `0Hz`. Default Value: `10M`
Frequency Max (Hz)	Sets the maximum frequency for the frequency sweep. Default Value: `10e9 or 10G`
Sampling Type	Specifies the frequency sampling type for the frequency sweep. Possible values are `Linear`, `Log`, and `Mixed`. Mixed type provides combined log sampling (at low frequencies) and linear sampling (at high frequencies). Default Value: `Linear`
Log Points per Decade	Specifies the log points per decade for log sampling. This field is enabled only when the *Sampling Type* is set to `Log` or `Mixed`. Default Value: `10`
Transition Frequency (Hz)	Specifies the transition frequency for switching between log sampling and linear sampling. This field is enabled only when *Sampling Type* is set to `Mixed`. Default Value: `10e6`
Linear Frequency Step Size (Hz)	Specifies the step size for linear sampling. This field is enabled only when *Sampling Type* is set to `Linear` or `Mixed`. Default Value: `10e6`
Solver Options	This section provides the option to specify how to run the solver.
Max Number of CPU to use in simulation	Specifies how many CPU cores you want to use to run the simulation. This setting is used by the EMX solver only. Default value: `0`, which implies that the solver automatically assigns the most suitable number of CPUs.
Remote Job Execution	This section displays the option for job distribution run by the simulator.
Prefix Solver Commands with	Displays the DRMS command set for distributed processing of simulation runs to multiple resources. Environment variable: drmsCommand

© 2023 Cadence Design Systems, Inc.

The Clarity Tab in the Simulation Settings Form

The following table describes the settings used by Clarity 3D Solver.

Field	Description
Adaptive Solution	This section provides options to specify frequency settings to be used by the simulator
Solution Frequency (Hz)	Specifies the frequency at which the adaptive solution is performed. This is typically set to *Frequency Max*. Default Value: `10G`
Max Number of Adaptive Mesh Iterations	Specifies the maximum number of mesh refinements that can be performed. Adaptive meshing stops when the specified number of iterations are complete or Target Delta S is attained, whichever comes first. Default Value: `50`
Adaptive Refinement Percentage (%)	Specifies the percentage of unknowns acceptable in refinement. In each mesh refinement iteration, an increase in number of unknowns is less than the specified percentage of existing number of unknowns. Default Value: `10`
Target Delta S	Specifies the target s-parameter difference to be attained. Adaptive meshing stops when the target difference is attained or the maximum number of adaptive mesh iterations are complete, whichever comes first. Default Value: `0.02`
Min Number of Adaptive Iterations	Specifies the minimum number of mesh refinements to be performed. Adaptive meshing does not stop until the refinement pass is equal or greater than this number. Enter a number equal to or greater than `3`. However, `3` is used if a number less than `3` is entered. Default Value: `1`
Min Number of Converged Iterations	Specifies the number of times the convergence criterion must be met consecutively for the adaptive mesh algorithm to converge. For example, if this is set to `2`, adaptive meshing does not stop until the convergence criterion is met twice in a row. Default Value: `1`

© 2023 Cadence Design Systems, Inc.

Field	Description
Solver Options	This section provides the settings that specify how to run the solver.
Metal Type	Specifies the metal type to be used. Possible values are: ■ `Metal_Inside`: Model metals with elements inside metals. This type is least memory efficient. It provides more accurate low frequency resistance. It also provides a different solution as compared to `Metal_Skin_Impedance` at high frequency range. This type is recommended when Explicit DC is needed. ■ `Metal_Skin_Impedance`: Model metals with a frequency dependent skin effect impedance boundary conditions on the exterior surfaces of the metals. This type provides very accurate high frequency resistive loss. This type is recommended for most applications. ■ `Auto_Fitting`: Model metals with a frequency dependent skin effect impedance boundary conditions on the exterior surfaces of the metals. This type provides very accurate high frequency resistive loss. This type is recommended for most applications. ■ `DC_Thickness`: Model metals with a frequency dependent skin effect impedance boundary conditions on the exterior surfaces of the metals. This type provides very accurate high frequency resistive loss. This type is recommended for most applications. Default Value: `Metal_Inside` for IC layouts, `Metal_Skin_Impedance` for package layouts

© 2023 Cadence Design Systems, Inc.

Field	Description
Basis Function Order	Describes the mesh element polynomial type. Possible values are: ■ `ZERO`: Uses 0th order elements with linear E-fields inside the element and constant E-fields along an edge. This uses less memory than the 1st order with the same number of elements, but requires finer (more) elements to achieve solution convergence. ■ `FIRST`: Uses high order elements with high order polynomial E-fields inside the element and along an edge. This uses more memory than 0thorder with the same number of elements, but requires fewer elements to achieve solution convergence. Default Value: `FIRST`
Matrix Solver	A label that is always set to `Automatic`, which implies that the solver automatically switches to out-of-core (OOC) solvers when necessary.
Port De-embedding	Removes the parasitic inductance introduced by the ports. This option is applicable to rectangular lumped ports only. When you select this setting, Clarity returns the *modelName*_`deembedded.SnP` file to Layout MXL.
Geometry Options	This section displays the option for job distribution run by the simulator.
Meshing Algorithm	This field is set to `DMesh`, which implies that Clarity uses an MCAD meshing process to generate the material and the `.w3d` files. The generated mesh is coarser in nature.
dz+ (in um)	Specifies the thickness of the top air buffer, which is the distance (buffer zone size) between the simulation region (outer box) and the design (inner box) on the +Z axis. Default Value: `1000`
dz+ (in um)	Specifies the thickness of the bottom air buffer, which is the distance (buffer zone size) between the simulation region (outer box) and the design (inner box) on the -Z axis. Default Value: `100`

© 2023 Cadence Design Systems, Inc.

Field	Description
um	Specifies the condition for *dz+* or *dz-*. Possible values are: ■ `Perfect Electrical Conductor` ■ `Perfect Magnetic Conductor` ■ `Approximately Open` Default Value: `Approximately Open` (for both *dz+* and *dz-*)
Signal Net Max Edge Length	Specifies the maximum length for the edges of triangles used to create the surface mesh on signal nets
Buffer Size	The simulation region in Clarity is larger than the geometry bounding box defined by minimum and maximum x-y-z dimensions of the design. The settings in this section define distances (buffer zone size) between the simulation region (outer box) and the design (inner box). As a rule of thumb, the buffer zone size should be 5-10 times of the distance between top and bottom metal layers.
dx+	Specifies the distance (buffer zone size) between the simulation region (outer box) and the design (inner box) on the +X axis. Default Value: `1000`
dx-	Specifies the distance (buffer zone size) between the simulation region (outer box) and the design (inner box) on the -X axis. Default Value: `1000`
dy+	Specifies the distance (buffer zone size) between the simulation region (outer box) and the design (inner box) on the +Y axis. Default Value: `1000`
dy-	Specifies the distance (buffer zone size) between the simulation region (outer box) and the design (inner box) on the -Y axis. Unit: um Default Value: `1000`

© 2023 Cadence Design Systems, Inc.

Field	Description
Conformal Outer Box	Used to enforce the exterior boundary to be the same as the metal shape profile in the XY plane. If this check box is selected, the side walls of the outer box are determined by the enabled cutting polygon. As Clarity targets non-radiation structures, and at a certain distance, the field decays almost to zero, the boundary distance from the structure must be large enough to allow fields to decay to zero.
Dielectric Buffer Size	If set to `0`, the outer box side walls are coincident with the enabled cutting boundary. When the Outer Box Boundary condition is set to `ABC`, it approximates an open region that extends to infinity, and removes the truncation boundary reflections. If set to a value greater than `0`, the outer box side walls are based on the enabled cutting boundary and are expanded by this value. Default Value: `0` This field overrides all the separate wall conditions that you have specified in this section. For example, if `Approximately Open` is selected in the *Boundary Conditions* drop-down list box, the same value is used for all the side walls of the outer box.
Drop-down list for all controls in this group	Specify how to model the outer box surfaces. Possible values are: ■ `Perfect electrical conductor` ■ `Perfect magnetic conductor` ■ `Approximately open` Default Value: `Perfect Electrical Conductor` for IC designs, `Approximately Open` for package designs

The EMX Tab in the Simulation Settings Form

The following table describes the settings used by EMX 3D Planar.

Field	Description
Mesh	This section contains the options to control the geometry of the mesh EMX creates for the model.

© 2023 Cadence Design Systems, Inc.

Field	Description
Edge Mesh	Specifies the width of the mesh elements at the edge of the conductors. Default is `1.000 microns`
Thickness	Specifies the maximum thickness allowed for conductors. If the conductors are thicker than this value, they are split vertically. Default value: `1.000 microns`
Via Merge	Controls via merging at a global level. To specify a common distance criteria to merge vias globally, set the minimum allowed distance in this field. Vias that are closer than the given distance are merged. If you are controlling vias on a layer-by-layer basis in the process setup, set this field to `0`. Default value: `0`
3D Metals	Specifies a list of metal names for which EMX needs to use 3D models. By default, EMX uses 2D models for all metals. Specify a value in the following format: ■ To consider all metal conductors as 3D, specify `*`. ■ To use 3D models for a selected set of metal conductors, specify a comma-separated list of metal names. For example, `me7,me8,me9`. ■ To use 3D models for all metal conductors except a few, specify a comma-separated list of metal names prefixed with a hyphen (-). For example, `-me3,me4`. ■ To use 2D models for all metals, leave it blank. Default value: `*`

© 2023 Cadence Design Systems, Inc.

Field	Description
Via Capacitance	Specifies a list of vias for which you want EMX to model capacitance. The default value is a blank string, which specifies that for all vias, EMX need only model resistance. Specify a value in the following format: ■ To model capacitance for all vias, specify `*`. ■ To consider the capacitive effects for selected vias, specify their names in a comma-separated list. For example, `via3,via4`. ■ To include the capacitive effects for all vias except a few, specify a list of the names of vias to be excluded prefixed with a hyphen (-). For example, `-via3,via4` **Note:** EMX always converts names to lowercase. Therefore, it is recommended to use lowercase when specifying names for components in your layout.
Via Inductance	Specifies a list of vias for which you want EMX to model inductance. The default value is a blank string, which specifies that EMX does not include inductance for all vias. To consider the inductive effects for one or more vias, specify a list of the names of vias in the following format: ■ To model inductance for all vias, specify `*`. ■ To consider the inductive effects for selected vias, specify their names in a comma-separated list. For example, `via3,via4`. ■ To include the inductive effects for all vias except a few, specify a list of the names of vias to be excluded prefixed with a hyphen (-). For example, `-via3,via4`. **Note:** EMX always converts names to lowercase. Therefore, it is recommended to use lowercase when specifying names for components in your layout.
Solver	This section provides an option for the solver.
Full Wave (radiation)	Specifies whether to model radiation effects. Correct modeling of high-frequency substrate losses with highly-conductive substrates (around 1000 S/m) requires modeling with radiation.
Save Currents For Single Frequency Simulations	Informs the EMX simulator to save currents during the simulation. Currents are saved only for simulations at a single frequency point. This option is ignored for frequency sweeps.

© 2023 Cadence Design Systems, Inc.

Field	Description
Advanced	This section contains fields where you can specify command-line arguments for the advanced options to be used by EMX.
EMX+GDSView	Specifies a list of command-line arguments to be used for both EMX and GDS view, a 2D preview window for models that are using EMX. Use the `--definitions-file` command-line option to specify the definitions file for EMX. The file path must to be an absolute path and it can contain shell environment variables. For example, `--definitions-file $PROJECT/gpdk90def.proc`.
EMX	Specifies a list of command-line arguments to be used only for EMX.
GDSView	Specifies a list of command-line arguments to be used only for GDSview.

The LVS Tab in the Simulation Settings Form

The following table describes the settings used by the LVS flow.

Field	Description
Extraction Settings	This section contains the options for Quantus extraction.
Quantus Parasitic Blocking File	Specifies the `Quantus hcell_qci` file when `MOMCAPs` are blocked for extraction.
Convert [] to <>	Converts [] bus bit characters to <> when LVS is run with [] as characters because Virtuoso uses <> as bus bit characters.
Port Generation Settings	This section contains the options to merge or convert ports in the LVS flow.
Merge ports of multiplied instances	Merges all layout instances corresponding to a schematic instance to a single port. This option applies only to instances that have a multiplier specified in the schematic. By default, each layout instance gets a unique port.
Port width for top-level pin [um]	Makes square internal port of the size specified in this field if a top-level pin has no pin shape.

© 2023 Cadence Design Systems, Inc.

Field	Description
Convert edge ports to internal ports	Specifies whether to convert edge ports to internal ports if multiple edge ports overlap.
Internal port width to height ratio	Specifies the height ratio of the ports when converting them to internal ports. A value of `0.1` means the port width will be a tenth of its height, for example, a port with height `1um` will be `0.1um` wide.
Internal port minimum width [um]	Specifies the minimum width to use when converting edge ports to internal ports. If the port width to height ratio results in a very narrow port, this minimum width is used instead.

Related Topics

Specifying Shape Simplification Options for IC Layouts

Electromagnetic Solver Assistant Forms

© 2023 Cadence Design Systems, Inc.

Via Clustering Settings Form

Use the Via Clustering Settings form to configure options for via clustering. Via clustering requires the maximum spacing and maximum number of cuts for each via layer.

Field	Description
Cut Layer	Specifies the names of via layers found in the layout
Max Spacing	Specifies the spacing (in microns) between cuts. Any cuts that are within the given spacing are considered for clustering. Set this value to greater than or equal to cut spacing to cluster the vias on a layer.
Max Cuts	Specifies the maximum array size for a cluster. For example, a value of 50 would create a single shape up to a 50 x 50 array. This is typically set to a large number so that all cuts in a via are clustered.

Related Topics

Configuring Settings for Electromagnetic Simulations

Running an EMX Simulation (IC Layouts Only)

Running a Simulation using Clarity 3D Solver

Specifying Options for Via Clustering

Saving Default Parameters for Via Clustering

© 2023 Cadence Design Systems, Inc.

© 2023 Cadence Design Systems, Inc.

C

Electromagnetic Solver Assistant Environment Variables

The environment variables described in this appendix are used to control the settings for the Electromagnetic Solver assistant in Virtuoso Layout MXL:

- Shell Environment Variables
- Variables to Customize Extracted View Creation
- Variables to Customize the Settings for Virtuoso 3D Viewer
- Variables to Customize the Settings of the Electromagnetic Solver Assistant
- Variables to Customize the Settings for the Models in IC Design Layouts
- Variables to Customize the Settings for the Models in Package Design Layouts
- Variables to Customize the Settings in the LVS Flow
- Variables for the Custom Passive Device Authoring Flow

© 2023 Cadence Design Systems, Inc.

Shell Environment Variables

While launching the Electromagnetic Solver assistant, the tool looks for the solvers in the paths set by the PATH environment variable or the solver-specific variables listed in this section:

- CDS_CLARITY_LICENSE
- CLARITY_PATH
- EMX_PATH
- GDSVIEW_PATH
- VEM_PDF_VIEWER

© 2023 Cadence Design Systems, Inc.

CDS_CLARITY_LICENSE

Specifies the default license suite for Clarity. Set this variable to avoid the display the Choose License Suites form when you need to use the same license suite every time Clarity is launched.

Example

```
setenv CDS_CLARITY_LICENSE <license-name>
```

Related Topics

Launching the Electromagnetic Solver Assistant

Shell Environment Variables

© 2023 Cadence Design Systems, Inc.

CLARITY_PATH

Sets the full path to the executable for Clarity 3D Solver. It is not sufficient to provide only the path to the installation hierarchy.

Example

```
setenv CLARITY_PATH path-to-Clarity-executable
```

Related Topics

Launching the Electromagnetic Solver Assistant

Shell Environment Variables

© 2023 Cadence Design Systems, Inc.

EMX_PATH

Sets the full path to the executable for the EMX solver. It is not sufficient to provide only the path to the installation hierarchy.

Example

```
setenv EMX_PATH path-to-EMX-executable
```

Related Topics

Launching the Electromagnetic Solver Assistant

Shell Environment Variables

© 2023 Cadence Design Systems, Inc.

GDSVIEW_PATH

Sets the full path to the executable for the GDSview model preview window used by EMX models. It is not sufficient to provide only the path to the installation hierarchy.

Example

`setenv GDSVIEW_PATH` *`path-to-gdsview`*

Related Topics

Launching the Electromagnetic Solver Assistant

Shell Environment Variables

© 2023 Cadence Design Systems, Inc.

VEM_PDF_VIEWER

Sets the full path to the PDF viewer in which the layer stackup for EMX models is displayed. It is not sufficient to provide only the path to the installation hierarchy. By default, this variable is set to the path for the Evince PDF viewer.

Example

```
setenv VEM_PDF_VIEWER path-to-pdfViewer
```

Related Topics

Viewing the Layer Stackup for a Model

Shell Environment Variables

© 2023 Cadence Design Systems, Inc.

Variables to Customize Extracted View Creation

This section describes the following environment variables that are used while creating an extracted view:

- couplingCapMode
- interpolationMethod

© 2023 Cadence Design Systems, Inc.

couplingCapMode

```
cfde.smartview couplingCapMode cyclic { "ground" | "position" }
```

Description

Specifies how to stitch a coupling capacitor when one of the two nets attached to the capacitor is included in the model. This variable is used when the reference view used for the extracted view is a Quantus Smart View.

Possible values are:

- `ground`: The coupling capacitor of the net excluded from the model is connected to ground. This is the default value.
- `position`: The coupling capacitor of the net excluded from the model is connected to the nearest surviving node of the net included in the EM model.

GUI Equivalent

None

Examples

```
envGetVal("cfde.smartview" "couplingCapMode")
envSetVal("cfde.smartview" "couplingCapMode" 'cyclic "position")
```

Related Topics

Creating Extracted Views from Models

Variables to Customize Extracted View Creation

© 2023 Cadence Design Systems, Inc.

interpolationMethod

```
cfde.spectre interpolationMethod cyclic { "default" | "linear" | "spline" |
    "bbspice" }
```

Description

Specifies the interpolation method to be used by Spectre when simulating S-parameter data specified in the nport. The value of this variable is reflected in the *Interpolation Method* field of the Edit Properties form for the modeled device.

Possible Values:

- `linear`: Uses linear interpolation to get a data point needed in the sample that is not directly in the S-parameter file.

  Note: If the S-parameter file represents an active device, specify `"linear"` interpolation, instead of `"default"` or `"bbspice"` because `"bbspice"` enforces passivity.

- `spline`: Uses a cubic spline algorithm. Cubic spline can occasionally introduce errors when there are rapid changes in the transfer functions defined in the S-parameter file near the sample point.

- `bbspice`: Uses a rational model to represent the S-parameter data. This method is used to do a rational fit.

- `default`: Allows Spectre to automatically use an interpolation method. Spectre uses the default value for the `interp` option of an nport according to the Spectre global option `nport_default_interp`. When `nport_default_interp` is set to `auto_switch`, the nport automatically switches the interpolation method depending on the analysis. It chooses `bbspice` for the pss shooting Newton analysis, and `linear` for other analyses, such as ac, dc, and sp.

  If an nport instance has the `interp` option explicitly specified, the value of `interp` for that instance takes priority over the global option `nport_default_interp`. For all nport instances in the netlist that do not have `interp` set explicitly, they use the value specified in `nport_default_interp`.

Default value: `default`

GUI Equivalent

None

© 2023 Cadence Design Systems, Inc.

Examples

```
envGetVal("cfde.spectre" "interpolationMethod")
envSetVal("cfde.spectre" "interpolationMethod" 'cyclic "linear")
```

Related Topics

Interpolation Method in Analog Library Reference

Variables to Customize Extracted View Creation

Also see `spectre -h nport` for information on how `nport_default_interp` works for the version of Spectre you are using

© 2023 Cadence Design Systems, Inc.

Variables to Customize the Settings for Virtuoso 3D Viewer

Use the following variables to configure the settings for Virtuoso 3D Viewer:

- backgroundColor
- defaultView
- dynamicWireframes
- lightEnabled
- netsHighlightColors
- panningMode
- displayOrientAxes
- displayPorts
- displayRotationAxes
- displayWireframes
- displaySurfaces
- viewStackMaxCapacity
- wireframesColor
- zScale

© 2023 Cadence Design Systems, Inc.

backgroundColor

`v3d backgroundColor string` *`colorCode`*

Description

Specifies the background color of the canvas in the 3D Viewer window. Any change in the value of this variable is applicable to all new windows opened in 3D Viewer.

The default value is `"#858585"`.

GUI Equivalent

None

Examples

```
envSetVal("v3d" "backgroundColor" 'string "#808080")
envGetVal("v3d" "backgroundColor")
```

Related Topics

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

defaultView

```
v3d defaultView cyclic { "top" | "bottom" | "left" | "right" | "front" | "back" |
     "isometricSE" | "isometricSW" | "isometricNE" | "isometricNW" }
```

Description

Specifies the default view type to display the model. Any change in the value of this variable is applicable to all new 3D Viewer windows opened after you set the variable.

You can change the view by using the commands on the View toolbar or the *View* menu of the 3D Viewer window.

The default value is `"isometricNE"`.

GUI Equivalent

None

Examples

```
envGetVal("v3d" "defaultView" )
envSetVal("v3d" "defaultView" 'cyclic "top")
```

Related Topics

View Menu of Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

dynamicWireframes

```
v3d dynamicWireframes boolean { t | nil }
```

Description

Dynamically shows or hides the wire frames during rotation or dynamic panning and shows them again after the task in progress is complete. This improves the performance of these tasks.

This variable is applicable to all 3D Viewer windows in the current session and takes effect only if displayWireframes is set to `t`.

The default value is `t`.

GUI Equivalent

None

Examples

```
envGetVal("v3d" "defaultView")
envSetVal("v3d" "dynamicWireframes" 'boolean nil)
```

Related Topics

Options Menu of Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

lightEnabled

```
v3d lightEnabled boolean { t | nil }
```

Description

Specifies whether to activate lights thrown on the canvas. Light is helpful in identifying edges when you hide wire frames.

The default value is `t`.

GUI Equivalent

Options menu – *Light*

Display Options toolbar – *Light*

Examples

```
envGetVal("v3d" "lightEnabled")
envSetVal("v3d" "lightEnabled" 'boolean t)
```

Related Topics

Options Menu of Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

netsHighlightColors

```
v3d netsHighlightColors string highlight_color_packet
```

Description

Specifies the packets to be used for highlighting markers in the Nets assistant. If a non-existing packet is specified, 3D Viewer issues a warning message in CIW and resets the environment variable to the default highlight packet list.

You can specify any custom packets you created in the Display Resource Editor as a value for this variable.

Note: Any change in the value of this variable is applicable to all new windows opened in 3D Viewer.

The default value is `"hilite hilite1 hilite2 hilite3 hilite4 hilite5 hilite6 hilite7 hilite8"`.

GUI Equivalent

None

Examples

```
envGetVal("v3d" "netsHighlightColors")
envSetVal("v3d" "netsHighlightColors" 'string "halo1 halo2 halo3 halo4")
```

Related Topics

Nets assistant in Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

panningMode

```
v3d panningMode cyclic { "3D" | "viewport" }
```

Description

Specifies the default panning mode for 3D Viewer.

The default value is `"3D"`.

GUI Equivalent

Display Options toolbar – *Panning*

Examples

```
envGetVal("v3d" "panningMode")
envSetVal("v3d" "panningMode" 'cyclic "viewport")
```

Related Topics

Options Menu of Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

displayOrientAxes

```
v3d displayOrientAxes boolean { t | nil }
```

Description

Specifies whether to display or hide orient axes on the canvas.

The default value is `t`.

GUI Equivalent

Options menu – *Display Orient Axes*

Display Options toolbar – Display Orient Axes

Examples

```
envGetVal("v3d" "displayOrientAxes")
envSetVal("v3d" "displayOrientAxes" 'boolean nil)
```

Related Topics

Options Menu of Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

displayPorts

```
v3d displayPorts boolean { t | nil }
```

Description

Specifies whether to display or hide port numbers for the model.

The default value is `t`.

GUI Equivalent

Options menu – *Display Ports*

Display Options toolbar – *Display Ports*

Examples

```
envGetVal("v3d" "displayPorts")
envSetVal("v3d" "displayPorts" 'boolean nil)
```

Related Topics

Reviewing the Mesh Created by EMX in Virtuoso 3D Viewer

Options Menu of Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

displayRotationAxes

```
v3d displayRotationAxes boolean { t | nil }
```

Description

Specifies whether to display or hide the rotation axes on the canvas.

The default value is `t`.

GUI Equivalent

Options menu – *Display Rotation Axes*

Display Options toolbar – *Display Rotation Axes*

Examples

```
envGetVal("v3d" "displayRotationAxes")
envSetVal("v3d" "displayRotationAxes" 'boolean nil)
```

Related Topics

Options Menu of Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

displayWireframes

```
v3d displayWireframes boolean { t | nil }
```

Description

Specifies whether to display or hide the edges of the mesh. You can set this variable to hide the edges when you want to view the solid metal.

The default value is `t`.

GUI Equivalent

Options menu – *Display Wire Frames*

Display Options toolbar – *Display Wire Frames*

Examples

```
envGetVal("v3d" "displayWireframes")
envSetVal("v3d" "displayWireframes" 'boolean nil)
```

Related Topics

Reviewing the Mesh Created by EMX in Virtuoso 3D Viewer

Options Menu of Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

displaySurfaces

`v3d displaySurfaces boolean { t | nil }`

Description

Specifies whether to display or hide surfaces of the nets and instances in the model. The default value is `t`.

GUI Equivalent

Options menu – *Display Surfaces*

Display Options toolbar – *Display Surfaces*

Examples

```
envGetVal("v3d" "displaySurfaces")
envSetVal("v3d" "displaySurfaces" 'boolean nil)
```

Related Topics

Options Menu of Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

viewStackMaxCapacity

`v3d viewStackMaxCapacity int` *maxCapacity*

Description

When you use the commands to pan, zoom, rotate, scale, or change the view type, the tool updates the display of the model. It also saves the previous view of the model. This variable specifies the maximum number of views the tool can save in the memory stack. The default value is `10`.

GUI Equivalent

None

Examples

```
envGetVal("v3d" "viewStackMaxCapacity")
envSetVal("v3d" "viewStackMaxCapacity" 'int 12)
```

Related Topics

Virtuoso 3D Viewer

Reviewing the Mesh Created by EMX in Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

wireframesColor

`v3d wireframesColor string` *colorCode*

Description

Specifies the wireframe color for the shapes displayed in the 3D Viewer window. Any change in the value of this variable is applicable to all new windows opened in 3D Viewer.

The default value is `"#000000"`.

GUI Equivalent

None

Examples

```
envGetVal("v3d" "wireframesColor")
envSetVal("v3d" "wireframesColor" 'string "#FF0000")
```

Related Topics

Virtuoso 3D Viewer

Reviewing the Mesh Created by EMX in Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

zScale

`v3d zScale int` *zScaleValue*

Description

Specifies the scaling factor by which you want to scale the mesh on the z-axis. The scaled z-axis allows you to clearly see the thin layers or to view the mesh between the layers.

The default value is `1`.

GUI Equivalent

View menu – *Z-Scale*

Examples

```
envGetVal("v3d" "zScale")
envSetVal("v3d" "zScale" 'int 4)
```

Related Topic

View Menu of Virtuoso 3D Viewer

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

Variables to Customize the Settings of the Electromagnetic Solver Assistant

The variables in this category are common for both IC and package layouts.

- clusterVias
- diegroundScale
- dielectricBlockOtherLayer
- dielectricBlockPermittivity
- dielectricBlockSourceLayer
- drmsCommand
- emxLayerMap
- emxMinInternalPortWidth
- emxProcessCornerDirectory
- emxProcessDefaultCorner
- exportCellName
- exportLibName
- exportSparamViewName
- portLayerLocation
- processCornerDirectory
- processDefaultCorner
- runDirectory
- setNotEmbedded
- simplifyToSides

© 2023 Cadence Design Systems, Inc.

clusterVias

```
vem clusterVias boolean { t | nil }
```

Description

Species whether or not to cluster the vias.

Examples

```
envGetVal("vem" "clusterVias")
envSetVal("vem" "clusterVias" 'boolean t)
```

Related Topics

Specifying Options for Via Clustering

Variables to Customize the Settings of the Electromagnetic Solver Assistant

© 2023 Cadence Design Systems, Inc.

diegroundScale

`vem dieGroundScale float f_scaleValue`

Description

Species the default scale value for the *Die Ground* field on the Setup tab of the Electromagnetic Solver assistant.

GUI Equivalent

Command	*Window – Assistants – Electromagnetic Solver*
Field	*Simulation Settings – Die Ground*

Examples

```
envGetVal("vem" "dieGroundScale")
envSetVal("vem" "dieGroundScale" 'float 0.2)
```

Related Topics

Simulation Settings Form

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

dielectricBlockOtherLayer

`vem dielectricBlockOtherLayer string` *`layerName`*

Description

Species the name of the other MIMCAP plate layers above or below the source layer for a dielectric block to be inserted.

Default value: `""`

GUI Equivalent

None

Examples

```
envGetVal("vem" "dielectricBlockOtherLayer")
envSetVal("vem" "dielectricBlockOtherLayer" 'string "layerName")
```

Related Topics

Validating Layer Settings for a Package Layout

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

dielectricBlockPermittivity

`vem dielectricBlockPermittivity float` *`permittivityLevel`*

Description

Species the permittivity to use for a dielectric block.

Default value is `0.0`

GUI Equivalent

None

Examples

```
envGetVal("vem" "dielectricBlockPermittivity")
envSetVal("vem" "dielectricBlockPermittivity" 'float 0.2)
```

Related Topics

Validating Layer Settings for a Package Layout

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

dielectricBlockSourceLayer

`vem dielectricBlockSourceLayer string` *`t_layerName`*

Description

Species the name of the source layer for a dielectric block to be inserted. The dielectric block has the same size as the shapes on this layer. For MIMCAP, this is one of the plate layers.

Default value: `""`

GUI Equivalent

None

Examples

```
envGetVal("vem" "dielectricBlockSourceLayer")
envSetVal("vem" "dielectricBlockSourceLayer" 'string "layerName")
```

Related Topics

Validating Layer Settings for a Package Layout

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

drmsCommand

`drmsCommand` *`command`*

Description

Specifies the DRMS (Distributed Resource Management System) command to be used by the EM solvers to distribute jobs to a workload job scheduling software, such as LSF or OpenLava.

GUI Equivalent

None

Examples

```
envGetVal("vem" "drmsCommand")
envSetVal("vem" "drmsCommand" 'string "bsub....")
```

Related Topic

Distributing Simulation Runs to Multiple Resources

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

emxLayerMap

`vem emxLayerMap string` *`t_pathToLayerMapFile`*

Description

Specifies the path to the GDS layer map file to be used when sending models to the EMX solver. By default, the tool uses the GDS layer map provided in the technology library.

GUI Equivalent

None

Examples

```
envGetVal("vem" "emxLayerMap")
envSetVal("vem" "emxLayerMap" 'string "../project/layermap1")
```

Related Topics

Environment Settings Form

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

emxMinInternalPortWidth

`vem emxMinInternalPortWidth float f_portWidth`

Description

Species a minimum width for internal ports. If a pin is narrower than the minimum width, an edge port is created even when there is no direction defined for edge. The default value is `0.0`, which means internal ports are created regardless of pin width. For example, setting `emxMinInternalPortWidth` means if a pin width is less than `0.004um`, access direction is ignored and an edge port is created. For any pin that is at least `0.004um` wide, an internal port is created, assuming it has no access direction.

GUI Equivalent

None

Examples

```
envGetVal("vem" "emxMinInternalPortWidth")
envSetVal("vem" "emxMinInternalPortWidth" 'float 0.004)
```

Related Topics

Port Management for EMX Models in IC Layouts

Variables to Customize the Settings of the Electromagnetic Solver Assistant

© 2023 Cadence Design Systems, Inc.

emxProcessCornerDirectory

`vem emxProcessCornerDirectory string` *`t_cornerPath`*

Description

Specifies the path to the directory where `.proc` files with process corner settings are saved. By default, this variable is set to `""` and the Electromagnetic Solver assistant looks for the files in `.cadence/dfII/Sigrity/corners`. You can specify an absolute path or a relative path to the process corner directory. For relative paths, the current working directory for Virtuoso is considered as the base directory.

GUI Equivalent

None

Examples

```
envGetVal("vem" "emxProcessCornerDirectory")
envSetVal("vem" "emxProcessCornerDirectory" 'string "../project/procCornerFiles")
```

Related Topics

Environment Settings Form

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

emxProcessDefaultCorner

`vem emxProcessDefaultCorner string` *`t_cornerName`*

Description

Specifies the name of the default corner to be used for a new model created in the Electromagnetic Solver assistant. Set this environment variable when there are multiple `.proc` process corner files in the directory specified by emxProcessCornerDirectory.

When set to the default, the tool selects the first corner name available in the drop-down list.

GUI Equivalent

None

Examples

```
envGetVal("vem" "emxProcessDefaultCorner")
envSetVal("vem" "emxProcessDefaultCorner" 'string "EMXcorner1")
```

Related Topics

Environment Settings Form

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

exportCellName

`vem exportCellName string` *`t_cellName`*

Description

Specifies the value of the *Cell* field in the Create S-Parameter View form. The default value is `%cell%`, which will be replaced with the layout cell name.

GUI Equivalent

Command	*Window – Assistants – Electromagnetic Solver*
Field	*Create S-Parameter View – Cell*

Examples

```
envGetVal("vem" "exportCellName")
envSetVal("vem" "exportCellName" 'string "%cell%_%model%")
```

Related Topics

Creating S-Parameter Cellviews for Models

Create S-Parameter View Form

© 2023 Cadence Design Systems, Inc.

exportLibName

`vem exportLibName string t_libName`

Description

Specifies the value of the *Library* field in the Create S-Parameter View form. The default value is `%lib%`, which will be replaced with the layout library name.

GUI Equivalent

Command	*Window – Assistants – Electromagnetic Solver*
Field	*Create S-Parameter View – Library*

Examples

```
envGetVal("vem" "exportLibName")
envSetVal("vem" "exportLibName" 'string "wLanPALib")
```

Related Topics

Creating S-Parameter Cellviews for Models

Create S-Parameter View Form

© 2023 Cadence Design Systems, Inc.

exportSparamViewName

`vem exportSparamViewName string *t_sparamviewName*`

Description

Specifies the value of the *View* field in the Create S-Parameter View form. The default value is `sparam_%model%`, which will be replaced with the layout view name.

GUI Equivalent

Command	*Window – Assistants – Electromagnetic Solver*
Field	*Create S-Parameter View – View*

Examples

```
envGetVal("vem" "exportSparamViewName")
envSetVal("vem" "exportSparamViewName" 'string "sparam_mmic")
```

Related Topics

Creating S-Parameter Cellviews for Models

Create S-Parameter View Form

© 2023 Cadence Design Systems, Inc.

portLayerLocation

```
vem portLayerLocation cyclic { "top" | "bottom" | ""}
```

Description

Specifies the location of same-layer ports. These ports can be placed on either the top edge or the bottom edge of the metal layers.

The default is `""`, which indicates that for the top most layer, same-layer ports are placed on the top edge. For all other layers, same-layer ports are placed on the bottom edge of the metal.

GUI Equivalent

None

Examples

```
envGetVal("vem" "portLayerLocation")
envSetVal("vem" "portLayerLocation" 'cyclic "top")
```

Related Topic

Specifying Ports for Clarity Models in Package Layout

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

processCornerDirectory

`vem processCornerDirectory string t_cornerDirPath`

Description

Specifies the path to the directory where `.emproc` files with process corner settings are saved. When this variable is set to `""`, the Electromagnetic Solver assistant looks for the files at `.cadence/dfII/Sigrity/corners`. You can specify an absolute path or a relative path to the process corner directory. For relative paths, the current working directory for Virtuoso is considered as the base directory.

GUI Equivalent

None

Examples

```
envGetVal("vem" "processCornerDirectory")
envSetVal("vem" "processCornerDirectory" 'string "../project/processCornerFiles")
```

Related Topics

Environment Settings Form

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

processDefaultCorner

`vem processDefaultCorner string`

Description

Specifies the name of the default corner to be used for a new model created in the Electromagnetic Solver assistant. Set this environment variable when there are multiple `.emproc` process corner files in the directory specified by processCornerDirectory.

When this variable is set to `""`, the tools selects the first corner name available in the drop-down list.

If the corner specified by this variable is not found, the EM Solver assistant searches for other `.emproc` files in the process corner directory and uses the first found process corner in the alphabetical order.

GUI Equivalent

None

Examples

```
envGetVal("vem" "processDefaultCorner")
envSetVal("vem" "processDefaultCorner" 'string "corner1")
```

Related Topics

Environment Settings Form

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

runDirectory

`vem runDirectory string` *`path_to_runDir`*

Description

Specifies the run directory for all the models created in the Electromagnetic Solver assistant. The tool also saves the `.clf` file for models at this location. You can use SHELL environment variables in the path specified by this variable.

By default, this variable is set to "", which specifies that the *`Virtuoso-current-working-directory`*`/.cadence/dfII/Sigrity` path is used as the run directory.

Note: If you have already created models for a layout before setting this variable, the model files saved at the previous location become unavailable if the new location is different.

GUI Equivalent

None

Examples

```
envGetVal("vem" "runDirectory")
envSetVal("vem" "runDirectory" 'string "$PROJECTDIR/models")
```

Related Topics

Environment Settings Form

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

setNotEmbedded

`vem setNotEmbedded string` *`t_listOfCellviews`*

Description

Specifies a list of cellviews whose instances are not to be considered as embedded components. Any instance of the specified cellviews is excluded from the extracted view even if it is selected on the *Selection* tab of the Electromagnetic Solver assistant.

This variable is particularly helpful when you have excluded layers from a model and want to exclude the instances embedded on those layers from the extracted view.

If an instance of the cellview listed in the value of this environment variable is selected for inclusion in the model and it has pins, the tool generates ports on its pins.

GUI Equivalent

None

Examples

```
envGetVal("vem" "setNotEmbedded")
envSetVal("vem" "setNotEmbedded" 'string "(myLib myCell myView)")
```

Related Topics

excludeLayers

Creating Extracted Views from Models

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

simplifyToSides

`vem simplifyToSides int` *`x_numPoints`*

Description

Specifies the number of points or sides for polygons to which circle and ellipse shapes in an IC layout are to be converted during the creation of a .clf file. By default, all circles and ellipses found in an IC layout are converted to hexagons. Use this variable to specify a different number of sides. Simplified shapes with fewer sides reduce the number of unknowns in the electromagnetic mesh to be used by AXIEM, and thereby, improve the performance of electromagnetic simulations.

GUI Equivalent

None

Examples

```
envGetVal("vem" "simplifyToSides")
envSetVal("vem" "simplifyToSides" 'int 8)
```

Related Topics

Layout Customization Options

Variables to Customize the Settings for Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

Variables to Customize the Settings for the Models in IC Design Layouts

You can use the following variables to customize the simulation settings for the models to be created for IC design layout:

- defaultSettingsFile
- defaultSimulator

© 2023 Cadence Design Systems, Inc.

defaultSettingsFile

`vem.ic defaultSettingsFile string` *`t_filePath`*

Description

Specifies the path to a settings file that you can use to override the default values for the fields in the Simulation Settings form for a design of IC fabric type.

GUI Equivalent

None

Examples

```
envGetVal("vem.ic" "defaultSettingsFile")
envSetVal("vem.ic" "defaultSettingsFile" 'string "~/proj1/myICSettingsFile")
```

Related Topics

Configuring Settings for Electromagnetic Simulations

Simulation Settings form

Variables to Customize the Settings for the Models in IC Design Layouts

© 2023 Cadence Design Systems, Inc.

defaultSimulator

```
vem.ic defaultSimulator cyclic { "Clarity" | "EMX" }
```

Description

Specifies the default simulator to bet set for the models created for an IC design layout.

Possible values are `"Clarity"` and `"EMX"`.

The default value is `"EMX"`.

GUI Equivalent

None

Examples

```
envGetVal("vem.ic" "defaultSimulator")
envSetVal("vem.ic" "defaultSimulator" 'cyclic "Clarity")
```

Related Topics

Creating a Model

Variables to Customize the Settings for the Models in IC Design Layouts

© 2023 Cadence Design Systems, Inc.

Variables to Customize the Settings for the Models in Package Design Layouts

The following variables are for a package design layout:

- defaultSettingsFile
- defaultSimulator
- discretizationMaxError

© 2023 Cadence Design Systems, Inc.

defaultSettingsFile

`vem.package defaultSettingsFile string` *`filePath`*

Description

Specifies the path to a settings file that you can use to override the default values for the fields in the Simulation Settings form for a design of package fabric type.

GUI Equivalent

None

Examples

```
envGetVal("vem.package" "defaultSettingsFile")
envSetVal("vem.package" "defaultSettingsFile" 'string "~/proj1/myPkgSettingFile")
```

Related Topics

Configuring Settings for Electromagnetic Simulations

Simulation Settings form

Variables to Customize the Settings for the Models in Package Design Layouts

© 2023 Cadence Design Systems, Inc.

defaultSimulator

```
vem.package defaultSimulator cyclic { "Clarity" }
```

Description

Specifies the default simulator to bet set for the models created for a package design layout.

The default value is `"Clarity"`.

GUI Equivalent

None

Examples

```
envGetVal("vem.package" "defaultSimulator")
envSetVal("vem.package" "defaultSimulator" 'cyclic "Clarity")
```

Related Topics

Creating a Model

Variables to Customize the Settings for the Models in Package Design Layouts

© 2023 Cadence Design Systems, Inc.

discretizationMaxError

`vem.package discretizationMaxError float` *`f_maxErrorLimit`*

Description

Specifies the maximum acceptable error limit to be considered during discretization of the curved shapes in a package layout to polygons, which is done before sending the layout details to the EM solver. This error limit value is used by the Electromagnetic Solver assistant to identify the number of points for each polygon. For a high acceptable error limit, which also means a high error tolerance, each polygon shape has fewer points. As a result, the shape does not resemble the original shape. However, if the acceptable error limit is very low, each polygon has many points so that the resulting shape closely resembles the original shape it represents.

By default, the Electromagnetic Solver assistant takes the maximum acceptable error from the constraints set for the layout. It then prints the constraint name and the derived error limit in the CIW. You can use this variable to override the error limit automatically derived by the tool.

The default value of this variable is `0.0`, which implies that a value of 1/8th of the minimal spacing constraint value across all package spacing constraints in the foundry constraint group is used.

Note: If your layout does not contain any constraint, it is mandatory to set this environment variable.

GUI Equivalent

None

Examples

```
envGetVal("vem.package" "discretizationMaxError")
envSetVal("vem.package" "discretizationMaxError" 'float 1.0)
```

RelatedTopics

Variables to Customize the Settings for the Models in Package Design Layouts

© 2023 Cadence Design Systems, Inc.

Variables to Customize the Settings in the LVS Flow

The following variables are for settings in the LVS Flow:

- lvsLayerMapFile
- quantusLayerSetupFile
- quantusTrpFile
- svdbDirectoryLabelText

© 2023 Cadence Design Systems, Inc.

lvsLayerMapFile

`vem lvsLayerMapFile string` *`fileName`*

Description

Specifies the location of the LVS layer map file. This file is required when using LVS-based models because it contains the mapping from LVS layers to Virtuoso layers. The default value is `""`.

GUI Equivalent

The *LVS Layer Map* field in Environment Settings form

Examples

```
envGetVal("vem" "lvsLayerMapFile")
envSetVal("vem" "lvsLayerMapFile" 'string "~/proj1/myLayerMapFile")
```

Related Topics

Extracting Nets with EMX Using the LVS Flow

quantusLayerSetupFile

quantusTrpFile

© 2023 Cadence Design Systems, Inc.

quantusLayerSetupFile

`vem quantusLayerSetupFile string` *`fileName`*

Description

Specifies the path to the `layer_setup` file in the Quantus enablement. This file is part of the PDK and does not need to be created by you and it is needed only when extracting partial nets. The default value is `""`.

GUI Equivalent

The *Quantus Layer Setup File* field in Environment Settings form

Examples

```
envGetVal("vem" "quantusLayerSetupFile")
envSetVal("vem" "quantusLayerSetupFile" 'string "~/proj1/myQTLayerMapFile")
```

Related Topics

Extracting Nets with EMX Using the LVS Flow

lvsLayerMapFile

quantusTrpFile

© 2023 Cadence Design Systems, Inc.

quantusTrpFile

`vem quantusTrpFile string` *`fileName`*

Description

Specifies the path to the Quantus TRP file. Not all PDKs have a TRP file. It is needed only when device pin names are different on LVS layers and Virtuoso layers. The default value is `""`.

GUI Equivalent

The *Quantus TRP File* field in Environment Settings form in Environment Settings form

Examples

```
envGetVal("vem" "quantusTrpFile")
envSetVal("vem" "quantusTrpFile" 'string "~/proj1/myQTTrpFile")
```

Related Topics

Extracting Nets with EMX Using the LVS Flow

lvsLayerMapFile

quantusLayerSetupFile

© 2023 Cadence Design Systems, Inc.

svdbDirectoryLabelText

```
vem svdbDirectoryLabelText string dirName
```

Description

Specifies the user-defined value of the *SVDB Directory* label. The default value is `SVDB Directory`.

GUI Equivalent

Command	*Window – Assistants – Electromagnetic Solver*
Field	*SVDB Directory*

Examples

```
envGetVal("vem" "svdbDirectoryLabelText")
envSetVal("vem" "svdbDirectoryLabelText" 'string "LVS Ouput Folder")
```

Related Topics

Extracting Nets with EMX Using the LVS Flow

lvsLayerMapFile

quantusLayerSetupFile

quantusTrpFile

© 2023 Cadence Design Systems, Inc.

Variables for the Custom Passive Device Authoring Flow

The following variables are used in the Custom Passive Device Authoring flow:

- auCdlViewName
- createQRCData
- devicesFilePath
- deviceMapFilePath
- runDeviceSignature

© 2023 Cadence Design Systems, Inc.

auCdlViewName

`vem.cpd auCdlViewName string` *`aucdlViewName`*

Description

Generates customizable `auCdl` view for custom passive device during device signature generation. The default value is `aucdl`.

GUI Equivalent

None

Examples

```
envGetVal("vem.cpd" "auCdlViewName")
envSetVal("vem.cpd" "auCdlViewName" 'string "auCdlViewName")
```

Related Topics

runDeviceSignature

createQRCData

devicesFilePath

deviceMapFilePath

Variables for the Custom Passive Device Authoring Flow

Authoring Custom Passive Devices

© 2023 Cadence Design Systems, Inc.

createQRCData

```
vem.cpd createQRCData boolean { t | nil }
```

Description

Selects the Create Quantus RC data SVDB check box in the LVS Run form to create Quantus data (SVDB) and creates files required to run Quantus. The default value is `t`. When set to `nil`, the Create Quantus RC data SVDB check box is deselected in the LVS Run form and no files are created.

GUI Equivalent

The *Create Quantus RC data SVDB* field in the LVS Run form. It represents one part of the task done by using this variable.

Examples

```
envGetVal("vem.cpd" "createQRCData")
envSetVal("vem.cpd" "createQRCData" 'boolean nil)
```

Related Topics

auCdlViewName

runDeviceSignature

devicesFilePath

deviceMapFilePath

Variables for the Custom Passive Device Authoring Flow

Authoring Custom Passive Devices

© 2023 Cadence Design Systems, Inc.

devicesFilePath

`vem.cpd devicesFilePath string` *`conf_file_path`*

Description

Generates marker shapes over the created device using logical operations. The default value is `""`.

GUI Equivalent

Command	*Window – Assistants – Electromagnetic Solver*
Field	*Device Authoring – Generate Marker Shapes*

Examples

```
envGetVal("vem.cpd" "devicesFilePath")
envSetVal("vem.cpd" "devicesFilePath" 'string "/workdir/devices.txt")
```

Related Topics

auCdlViewName

createQRCData

runDeviceSignature

deviceMapFilePath

Variables for the Custom Passive Device Authoring Flow

Authoring Custom Passive Devices

© 2023 Cadence Design Systems, Inc.

deviceMapFilePath

`vem.cpd deviceMapFilePath string` *`conf_file_path`*

Description

Refers to a file that has required inputs to generate device signatures, such as recognition layer, marker layer, pin mapping, and context layers. The default value is `""`.

GUI Equivalent

Command	*Window – Assistants – Electromagnetic Solver*
Field	*Device Authoring – Generate Device Signature*

Examples

```
envGetVal("vem.cpd" "deviceMapFilePath")
envSetVal("vem.cpd" "deviceMapFilePath" 'string "/workdir/MapDevices.txt")
```

Related Topics

auCdlViewName

createQRCData

devicesFilePath

runDeviceSignature

Variables for the Custom Passive Device Authoring Flow

Authoring Custom Passive Devices

© 2023 Cadence Design Systems, Inc.

runDeviceSignature

```
vem.cpd runDeviceSignature cyclic { "none" | "pegasus" }
```

Description

Generates `auCdl` view of custom device and adds unique label to the custom device, when set to `pegasus`. In addition, generates extraction rules and device signatures.

Possible values are:

- `none`: The flow is run until generating a unique label on the device and creating an `auCdl` view.
- `pegasus`: (Default) The entire flow is run and device signatures generated.

GUI Equivalent

None

Examples

```
envGetVal("vem.cpd" "runDeviceSignature")
```

```
envSetVal("vem.cpd" "runDeviceSignature" 'cyclic "none")
```

Related Topics

auCdlViewName

createQRCData

devicesFilePath

deviceMapFilePath

Variables for the Custom Passive Device Authoring Flow

Authoring Custom Passive Devices

© 2023 Cadence Design Systems, Inc.

D

Virtuoso 3D Viewer

Use the Virtuoso® 3D Viewer environment to review the 3D mesh created by the EMX simulator for your electromagnetic models.

This chapter contains the following sections:

- Launching the Virtuoso 3D Viewer
- The 3D Viewer GUI

© 2023 Cadence Design Systems, Inc.

Launching the Virtuoso 3D Viewer

The Virtuoso 3D Viewer is currently used only to view a mesh created by the EMX simulator for an electromagnetic model created in the Electromagnetic Solver assistant. Therefore, you can open the 3D Viewer only after creating a model for simulation.

To create and open mesh for a model:

➡ Choose *Open Simulator – Generate Mesh* on the toolbar of the Electromagnetic Solver assistant.

  This command creates a mesh and saves it in a `.vmesh` file and then it launches 3D Viewer to display the generated mesh.

  If the mesh already exists, use the *Open Simulator – View 3D Mesh* command on the toolbar to open the most recent `.vmesh` file in the 3D Viewer.

The 3D Viewer is available in Virtuoso by default. It uses a MESA Software OpenGL driver that is included within Virtuoso.

To enable the 3D Viewer:

- Set the CDS_USE_OPENGL shell environment variable before running Virtuoso. This variable enables OpenGL, which is required to open the 3D Viewer environment.
- [Optional] Set the CDS_AUTO_NATIVE_OPENGL shell environment variable only if you want to use your own Linux-based OpenGL. This variable enables OpenGL, which is required to open the 3D Viewer environment.
- [Optional] Ensure that your X11 server supports the GLX extension that enables 3D graphics.

  Use the `xdpyinfo` utility to check whether the GLX extension is enabled:

  ```
  $ xdpyinfo | grep GLX
  => GLX
  ```

  If the GLX extension is enabled, `xdpyinfo` returns `GLX`. If not, refer to the user manual of your X11 server environment to learn how to enable the GLX extension.

Related Topics

Reviewing the Mesh Created by EMX in Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

Using Virtuoso 3D Viewer with OpenGL Enabled on Exceed onDemand

Certain versions of the OpenText Exceed onDemand connectivity solution do not support the resumption of applications that use OpenGL.

If you suspend an Exceed client session on which Virtuoso is running with OpenGL enabled, and then resume that Exceed session, you might not be able to resume Virtuoso. To prevent this when OpenGL is enabled, it is recommended that you close all open Virtuoso sessions before suspending the Exceed onDemand client session.

To launch the 3D Viewer, run the following two commands on the toolbar of the Electromagnetic Solver assistant:

- *Open Simulator – Generate Mesh*: Creates a mesh by using EMX, saves it in a `.vmesh` file, and launches the 3D Viewer to display the generated mesh.
- *Open Simulator – View 3D Mesh*: Opens the most recent `.vmesh` file in the 3D Viewer.

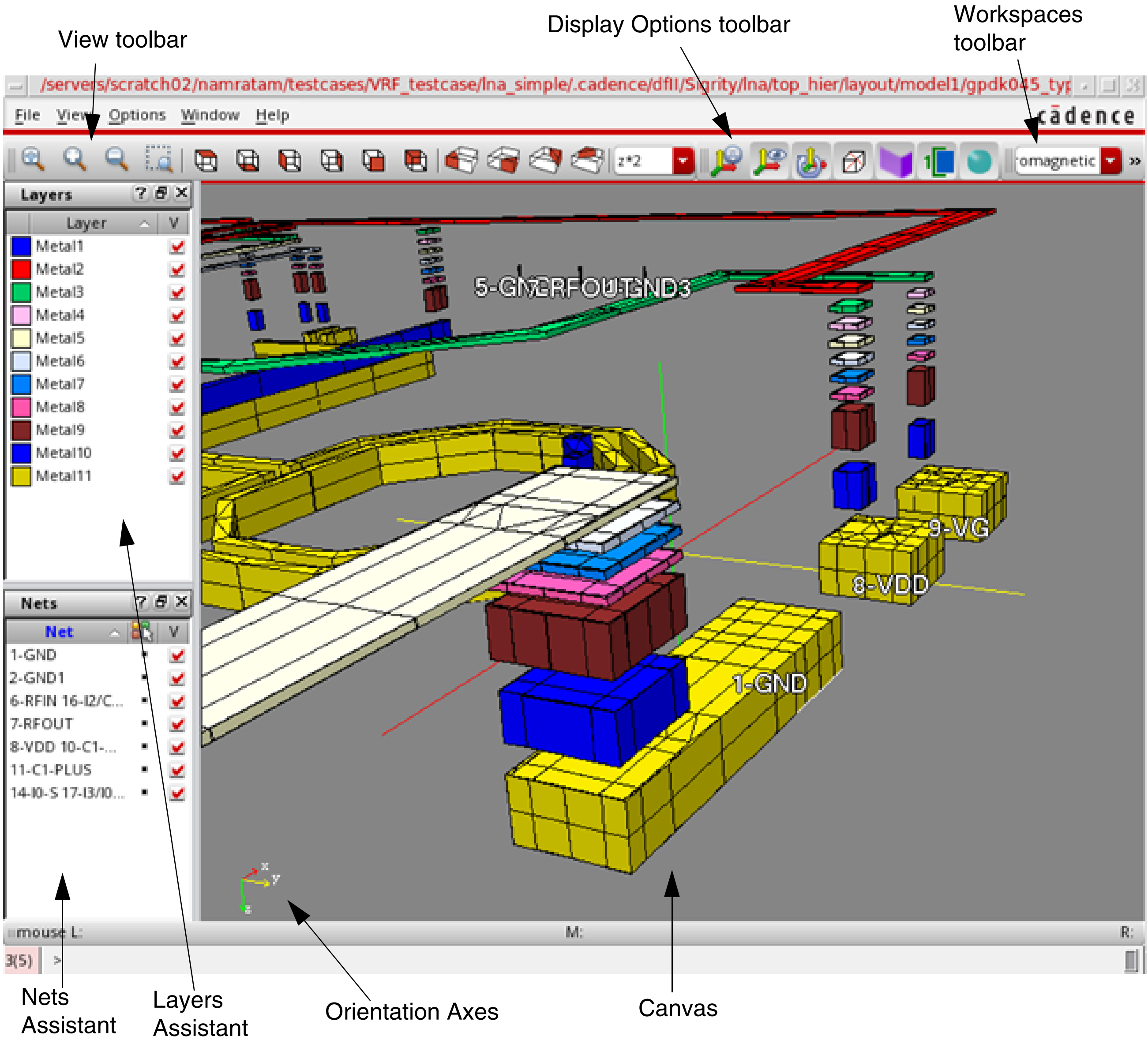
Related Topics

Reviewing the Mesh Created by EMX in Virtuoso 3D Viewer

© 2023 Cadence Design Systems, Inc.

The 3D Viewer GUI

The 3D Viewer graphical user interface contains the following elements:



- Canvas
- Menus
- Toolbars
- Assistants

© 2023 Cadence Design Systems, Inc.

Canvas

The canvas is the area where the mesh created by the solver is displayed. It shows all shapes and nets selected for the model.

By default, you see the isometric SE of the mesh, which appears to be in 2D, but you can click and hold the left mouse button and then slightly drag the pointer in the direction of your choice to rotate and view the 3D mesh. The orientation axes at the bottom left corner of the canvas shows the current direction of the x, y, and z axes providing guidance for rotation.

Use the left mouse button to rotate the view along the x and y axes. Press the `Ctrl` key and use the left mouse button to rotate the view along the x and z axes.

© 2023 Cadence Design Systems, Inc.

Menus

The 3D Viewer contains five menus described in this section.

Menu Command	Icon	Description
File		
Close		Closes the `.vmesh` currently displayed in the canvas.
Close All		Closes all `.vmesh` files currently displayed in the canvas.
View		
Zoom Fit		Fits the mesh in the given canvas area. Bindkey: `F`
Zoom In		Magnifies the mesh by a factor of 2. Bindkey: `Ctrl + Z`
Zoom Out		Reduces the mesh by a factor of 2. Bindkey: `Shift + Z`
Zoom to Area		Zooms the mesh view to a selected area. To select the rectangular area, move the pointer to the top-left corner of the area to zoom and right-click or press the `Z` bindkey. Drag the pointer to the bottom-right corner and release the mouse button to complete the selection.
View Top		Displays the top view of the model.
View Bottom		Displays the bottom view of the model.
View Left		Displays the left side view of the model.
View Right		Displays the right side view of the model.
View Front		Displays the front view of the model.
View Back		Displays the back view of the model.
Isometric SW		Displays the 3D view as if you are viewing it from the lower-left corner.

© 2023 Cadence Design Systems, Inc.

Isometric SE	Displays the 3D view as if you are viewing it from the lower-right corner. This is the default view.
Isometric NE	Displays the 3D view as if you are viewing it from the upper-right corner.
Isometric NW	Displays the 3D view as if you are viewing it from the upper-left corner.
Z-Scale	Scales the mesh on the z-axis by the selected factor. This command provides eight predefined scaling factors to choose from. The scaled z-axis allows you to clearly see the thin layers or to view the mesh between the layers.

Options

Display Orient Axes	Displays or hides the orientation axes in the canvas. Related environment variable: displayOrientAxes
Display Rotation Axes	Displays or hides the rotation axes in the canvas. Related environment variable: displayRotationAxes
Display Wire Frames	Displays or hides the edges of the mesh. Hide the edges when you want to view the solid metal. Related environment variable: displayWireframes
Display Surfaces	Displays or hides the surfaces. Hide the surfaces for a transparent mesh view with only wire frames. Related environment variable: displaySurfaces
Display Ports	Displays the ports in the mesh. Port numbers help you identify the ports corresponding to the ports in the *Ports* tab of the Electromagnetic Solver assistant. The selected ports are highlighted in a color to recognize them easily. Related environment variable: displayPorts

© 2023 Cadence Design Systems, Inc.

Light		Activates lights thrown on the canvas. This command is helpful in identifying edges when you hide wire frames. Related environment variable: lightEnabled
Window		
Assistants		Provides commands to show or hide dockable assistants.
Toolbars		Provides commands to show or hide toolbars.
Workspaces		This is the standard Virtuoso *Workspaces* menu command. For details, see Working with Workspaces.
Tabs		Provides commands to close the currently displayed or all the open tabs.
Help		This is the standard Virtuoso *Help* menu. For details, see Getting Help.

Tip

When you use the commands to pan, zoom, rotate, scale, or change the view type, the tool updates the display of the model. It also saves the previous view of the model. You can use the `W` and `Shift + W` bindkeys to open the previous or next view, respectively. Use the viewStackMaxCapacity variable to specify the maximum number of views the tool can save in the memory stack.

© 2023 Cadence Design Systems, Inc.

Toolbars

The 3D Viewer contains the following toolbars:

- View
- Display Options
- Workspaces

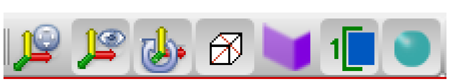
View

Provides functions to zoom the displayed mesh to the required level or to look at the mesh from different angles.

For details, see the View menu.

Display Options

Provides functions to display or hide the various components in the mesh view.



The first button on this toolbar is used to choose a panning mode to reposition the mesh in the viewer window. To pan, you can use the arrow keys on the keyboard. It repositions the mesh in the viewer window. Use it when you want to review the different parts of the mesh.

This toolbar button toggles between the following two modes:

	*Viewport Panning*	Pans the mesh in a two-dimensional space. In this mode, the entire window is shifted in the direction of the arrow key. The camera position is shifted but the direction in which it is pointing is not modified. The perspective and rotation point relative to the design location are not modified. If you zoom the view and pan the mesh, it appears that you are viewing a portion of the complete mesh by shifting the camera. Related environment variable: panningMode

© 2023 Cadence Design Systems, Inc.

3D Panning

Pans the mesh in a three-dimensional space. In this mode, panning does not change the direction in which the camera is pointing. The camera looks at the same point in the 3D space and the design is moved around that point. Both the perspective and the location of the rotation point relative to the design location are modified.

For example, if you start from fit view, the camera looks at the center of the window. If you perform 3D panning, the camera still looks at the center of the window, but the design location is moved.

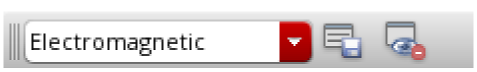
Related environment variable: panningMode

For details on the remaining buttons on this toolbar, see the Options menu.

Also see: wireframesColor

Workspaces

Provides functions to create or open a workspace. You can use a workspace to change the way you view your data by choosing what assistant panes and toolbars appear in your window, where they appear, and whether each assistant pane is docked or floating.



By default, the 3D Viewer shows the *Electromagnetic* workspace.

For more details on how to create your own workspace or to toggle assistants, see Working with Workspaces.

© 2023 Cadence Design Systems, Inc.

Assistants

The 3D Viewer contains three assistants:

- Layers
- Nets
- Currents Assistant

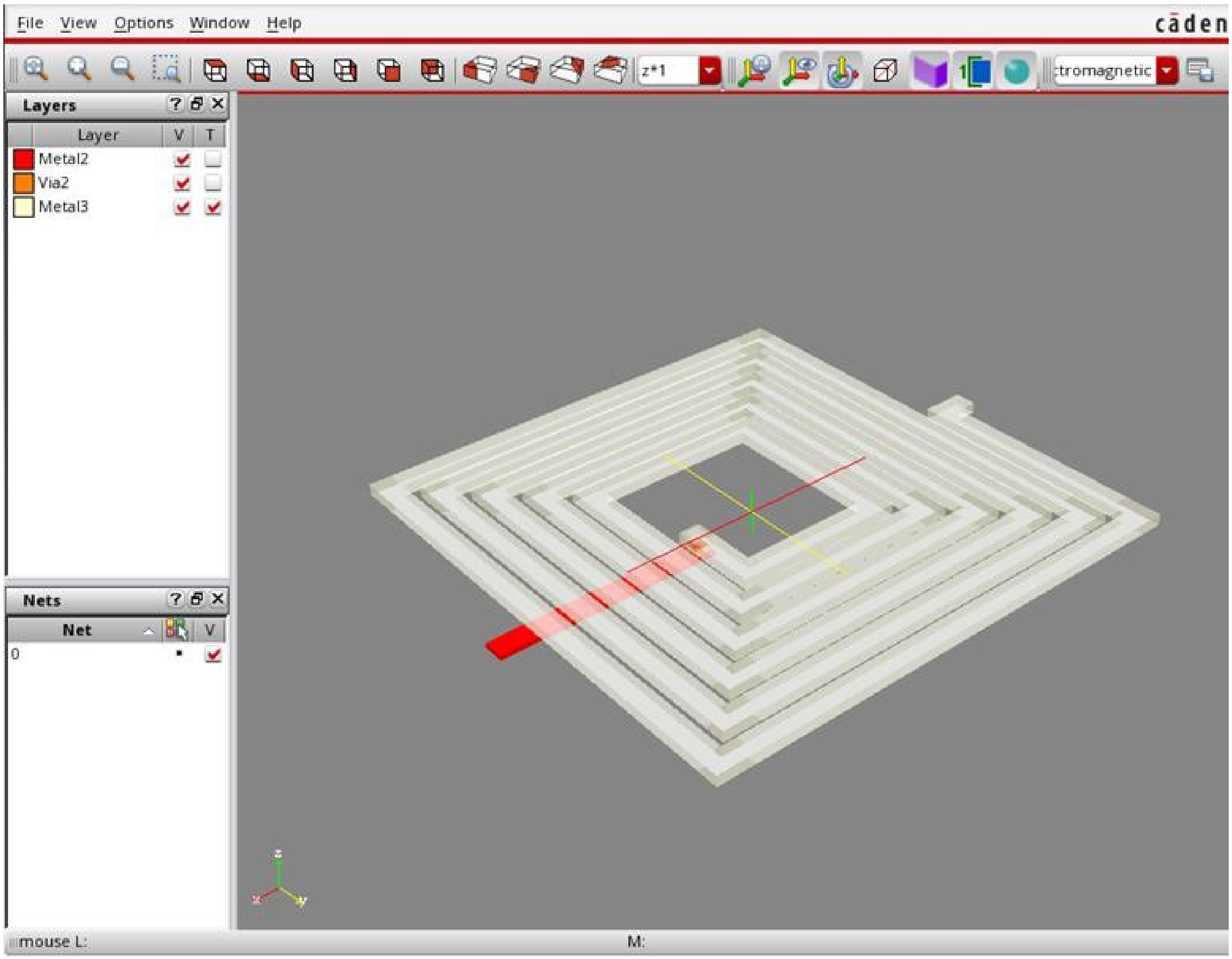
Layers Assistant

The Layers assistant provides controls to manage the visibility of layers in the mesh. When you open a mesh, the Layers assistant is displayed by default, docked in the left of the window.

The Layers assistant displays the names and colors of the layers on which the selected geometry objects are drawn in the layout view. The name and color of each layer is retrieved from the Palette assistant in Layout MXL.

© 2023 Cadence Design Systems, Inc.

If 3D Viewer is displaying layers from multiple libraries, all layers that belong to a common library are grouped together.

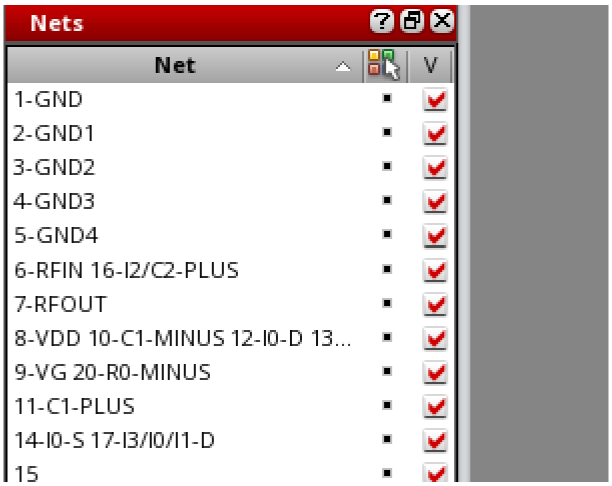


While reviewing a mesh, use the check box for a layer to control the visibility of that layer. This lets you to focus on specific layers and the shapes placed on them. To show or hide all layers of a library, select or clear the check box corresponding to the library name. You can also control the transparency of each layer in the Layers assistant of the Virtuoso 3D viewer by selecting the check boxes.

© 2023 Cadence Design Systems, Inc.

Nets Assistant

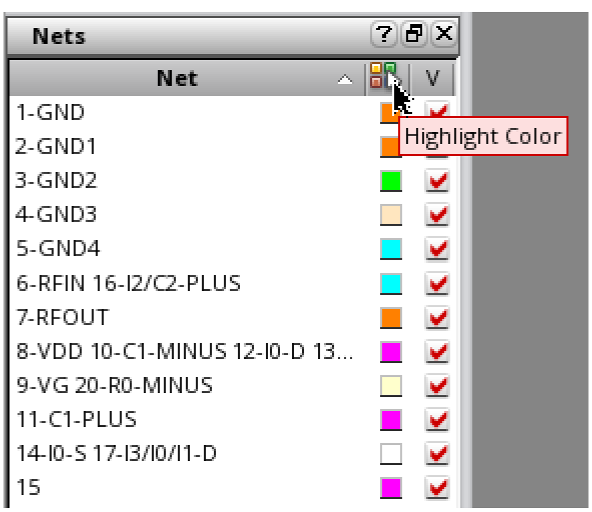
The Nets assistant lets you manage the visibility of nets and the metal islands electrically connected to them. When you open a mesh, the Nets assistant is displayed by default, docked below the Layers assistant.



The Nets assistant displays a list of nets connected to the shapes selected for the model. Use the check box to the right of each net to control the visibility of that net and the metal islands connected to it.

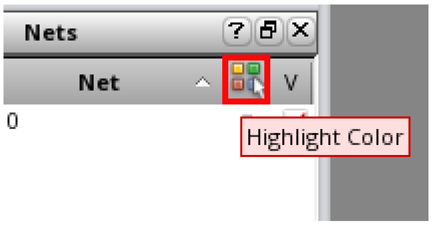
This assistant also lets you color the nets and the shapes connected to them in two modes:

- Use layer color: (Default) Uses the color of the layer on which the net is placed. In this mode, the color icon against each net is switched to *layerColor* mode.

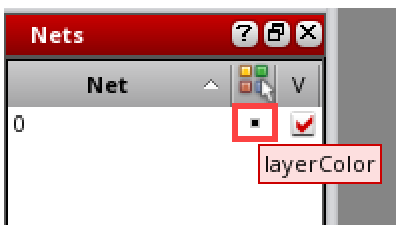


© 2023 Cadence Design Systems, Inc.

- Use net color: Uses a unique color for each net and the metal island connected to it. To enable this mode, click *Highlight Color* in the column header.



The 3D Viewer assigns a unique color to each net and displays that color next to the net name in the assistant. If all the available colors are used, some nets will be assigned the same colors. You can control the visibility of ports by selecting or deselecting the corresponding *Visibility* check box. The ports selected in the assistant are highlighted in red on the canvas.

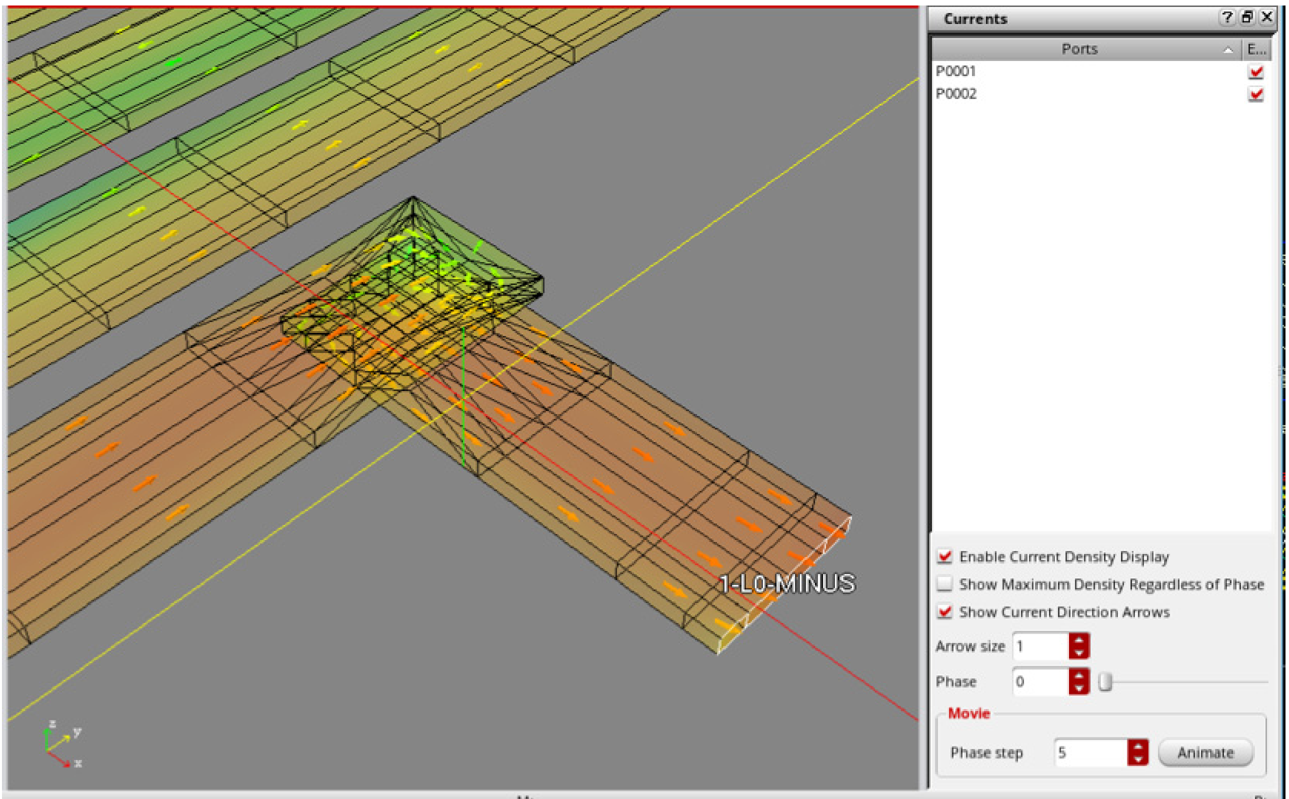


The color of the metal islands is also changed to match the color of the connected net.

© 2023 Cadence Design Systems, Inc.

Currents Assistant

The Currents assistant lets you manage the visibility of the current flow through the 3D structure. You can select an option while running EMX simulation to save current information to the VMESH file. Subsequently, the 3D viewer lets you view both the mesh and the current.



The Currents assistant displays a list of ports. Use the check box to the right of each port to control the voltage applied to it. The selected ports have voltage applied to it and the unselected ports are grounded.

You can choose from the following options to customize the view of the current flow:

- Select *Enable Current Density Display* to toggle between the classic mesh view and current view.
- Select *Show Maximum Density Regardless of Phase* to show the maximum current density at any phase value.
- Select *Show Current Direction Arrows* to display arrows to indicate the direction of current. *Arrow Size* lets you define the size of arrows.

© 2023 Cadence Design Systems, Inc.

- Use the *Phase* spin box or slider to show current density at a particular phase or time (`0` to `360`).
- Click the *Animate* button to start an animation automatically and to show the current at all phases over the time. Use *Phase step* to define how much phase to leap per half second.

© 2023 Cadence Design Systems, Inc.

E

Electromagnetic Solver Assistant SKILL Functions

This topic describes the public SKILL functions in the Virtuoso Electromagnetic Solver Assistant.

- cpdCreateDeviceInfo
- cpdGenerateDerivedShapes
- cpdLoadDerivedShapesRules
- cpdPegasusPreFormTrigger

© 2023 Cadence Design Systems, Inc.

cpdCreateDeviceInfo

```
cpdCreateDeviceInfo(
     [devMapFile t_devMapFile]
     [?cellView d_cellView]
     => t / nil
```

Description

Creates a label in the specified cellview as *libName_cellName_viewName* on the layer defined in Device Map file. It generates customizable `auCdl` view as defined in the argument. It creates a device signature and extraction rules for the custom passive device.

Arguments

`devMapFile` *t_devMapFile*

The path to the device layer mapping file. If it is not specified, the DeviceMap.txt is searched in current directory.

`?cellView` *d_cellView*

The cellview where the device information is created. If this is not specified, `geGetEditCellView()` is used by default.

Value Returned

`t`	The device information is created successfully.
`nil`	The device information could not be created.

Examples

```
cpdCreateDeviceInfo("DeviceMap.txt" ?cellView geGetEditCellView())
```

Gets the information required to generate device signature from the `DeviceMap.txt` file.

Related Topics

cpdGenerateDerivedShapes

cpdLoadDerivedShapesRules

© 2023 Cadence Design Systems, Inc.

cpdPegasusPreFormTrigger

Variables for the Custom Passive Device Authoring Flow

Authoring Custom Passive Devices

© 2023 Cadence Design Systems, Inc.

cpdGenerateDerivedShapes

```
cpdGenerateDerivedShapes(
     t_deviceName
     [?cellView d_cellView]
     [?devicesFile t_devicesFile]
     )
     => t / nil
```

Description

Generates new shapes in the specified cellview using the derived shape generation rules specified for a device in a device configuration file. If the file is not specified, the rules from the previously loaded device file (using `cpdLoadDerivedShapesRules`) are used.

Arguments

t_deviceName — The name of the device for which shapes are generated.

`?cellView` *d_cellView*

The cellview where shapes are generated. By default, it is the current cellview.

`?devicesFile` *t_devicesFile*

The name of the device configuration file that contains rules for shape generation.

Value Returned

`t`	The shapes are successfully generated.
`nil`	The shapes could not be generated.

Example

```
cpdGenerateDerivedShapes("Ind" ?cellView geGetEditCellView()  ?devicesFile
"devices.txt")
```

This generates a marker layer `instance drawing` over `IND2dummy`.

© 2023 Cadence Design Systems, Inc.

Related Topics

cpdLoadDerivedShapesRules

cpdCreateDeviceInfo

cpdPegasusPreFormTrigger

Variables for the Custom Passive Device Authoring Flow

Authoring Custom Passive Devices

© 2023 Cadence Design Systems, Inc.

cpdLoadDerivedShapesRules

```
cpdLoadDerivedShapesRules(
     t_configfileName
     )
     => l_devicenames / nil
```

Description

Loads the derived shapes generation rules into the memory of devices specified in the given file. The newly loaded list of rules overrides the previously defined list of rules for the devices.

Arguments

t_configfileName	The name of the file that contains the device generation rules.

Value Returned

l_devicenames	A list of devices for which the device generation rules have been updated.
nil	The list of devices could not be retrieved.

Examples

```
cpdLoadDerivedShapesRules("devices.txt")
```

The following is the format of `devices.txt`:

```
Devices(
  <deviceName>(
    (<derivedLayer> ( <operator> <arg1> [<arg2>] ) [<outputLayer> <outputPurpose>])
     ...
  )
  ...
)
```

Related Topics

cpdCreateDeviceInfo

cpdGenerateDerivedShapes

© 2023 Cadence Design Systems, Inc.

cpdPegasusPreFormTrigger

Variables for the Custom Passive Device Authoring Flow

Authoring Custom Passive Devices

© 2023 Cadence Design Systems, Inc.

cpdPegasusPreFormTrigger

```
cpdPegasusPreFormTrigger(
     t_runType
     t_presetFile
     x_windID
     [ ?userPreset t_userPreset ]
     )
     => l_devicenames / nil
```

Description

Pre-fill the Pegasus LVS form with custom passive device related configurations required to run LVS for custom passive devices.

Arguments

t_runType	Specifies the Pegasus utility that has been run. When you select *Pegasus – LVS*, this field gets defined automatically.
t_presetFile	The preset file used in Pegasus.
x_winId	The ID of the window from which the Pegasus LVS is launched.
?userPreset *t_userPreset*	
	Specifies if there is any saved preset file that you want to use apart from filling the form directly.

Value Returned

l_devicenames	The path of preset file that will be used to fill the Pegasus LVS form.
nil	The form could not be filled with the configuration information.

Example

You can use the following process to fill the Pegasus LVS form with custom passive device configuration settings if `PegausPreFormTrigger` is not already defined in your environment.

```
procedure(PegasusPreFormTrigger(runtype preset winId)
    cpdPegasusPreFormTrigger(runtype preset winId)
```


© 2023 Cadence Design Systems, Inc.

)

Related Topics

cpdCreateDeviceInfo

cpdGenerateDerivedShapes

cpdLoadDerivedShapesRules

Variables for the Custom Passive Device Authoring Flow

Authoring Custom Passive Devices

© 2023 Cadence Design Systems, Inc.