cadence®

Electromagnetic Solver Task Assistant

Product Version IC23.1
June 2023

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

Contents

1

Electromagnetic Solver Task Assistant

The Electromagnetic Solver assistant provides an interface to create 3D models for passive devices by running s-parameter extraction. The tool allows you to run full wave 3D Solvers, Clarity3D Solver, or EMX 3D planar solver.

You can perform the following tasks to create EM Models:

Configure Process Setup

- How to configure process settings for an IC layout?
- How to configure process setup for a package layout?

Define Models

- How to define a model for a full cellview?
- How to define a model for a layout-driven design?
- How to select nets and instances for models?
- How to create ports for models?
- How to simplify shapes?

Run Simulation

- How to configure simulation settings?
- What are the steps involved in running a simulation?

Create Extracted Views

- How to create an extracted view?

Related Content: Electromagnetic Solver Assistant User Guide

How to configure process settings for an IC layout?

For an IC layout, you need to provide process setup information. When using the Clarity 3D Solver simulator, specify the process setup information in the `.emproc` files that are saved in a process corner directory, which is by default set to the `.cadence/dfII/Sigrity/corners` directory in the current working directory. When using EMX, specify the process setup information in the `.proc` files. The EM Solver assistant reads the directory and loads the process corner names in the *Corner* column of the *Models* section. The default process corner directory used by EMX is `.cadence/dfII/Sigrity/corners`.

- Where to save the process settings for an IC layout?

When the layer stack information is available in an ICT or QRC technology file:

- Add a reference to the external ICT or QRC technology file along with the mapping information to map the external layers to Virtuoso layer.
- Provide details about substrates, height adjustment values for layers, values for dielectric simplification, and a list of layer-purpose pairs from which shapes are to be extracted.

Related Topics: Syntax of a corner referring to an external stackup

When the layer stack information is not available in an ICT or QRC technology file:

- Create a custom stackup with details about dielectric, metal, and via layers.
- Provide details about substrates, height adjustment values for layers, values for dielectric simplification, and a list of layer-purpose pairs from which shapes are to be extracted.

Related Topic: Syntax of a corner defining a custom stackup

How to configure process setup for a package layout?

For a package layout, the process setup information is read from the technology file and loaded in Virtuoso. If you have imported the layout from Allegro, the settings are also imported. You can validate the settings as described below.

Layer Stackup

1. Choose *RF - Module- Setup - Layer Stack Editor* to open the Layer Stack Editor form.
2. Check the values of thickness, conductivity, and permittivity of each metal and dielectric layer.

3. If required, modify these settings and click *OK*.

   Review the dielectric layer above and below the package layer. If needed, use the *Add Layer* command to add layer above and below it.

Related Topic: Layer Stack Editor

BGA Balls and Flip Chip Bump Setup

1. Select a BGA or IC instance.
2. Choose *RF - Module- Setup - Bump and Ball Editor* to open the Bump and Ball Editor form.
3. Validate the settings.
4. If required, modify these settings and click *OK*.

Related Topic: Bump and Ball Settings Editor

How to configure setup for a cross fabric design?

Cross-fabric extraction allows you to create a model across IC, package and board. It captures the full coupling between fabrics in a single 3D simulation. Each layer has its own models. The objective is to extract a model for the outer-most fabric, which is a board layout. Each fabric in the stack references the models of other fabrics placed within it.

To create a electromagnetic model for a cross fabric design:

1. Ensure that the prerequisites are met:
   - Each package and board fabric has a dielectric layer on top and bottom.
   - The height of the balls and bumps for each abstract instance equal to or higher than the height of the solder mask.

1. Open the outer-most fabric in VLS EXL and enable *Co-Design* mode.
2. Following the bottom-up approach, open the layout of the lowest fabric and define a model using the required objects from this layout.
3. Switch to the layer above the previous layer.
4. Create a new model by selecting objects from this layer and adding a reference to the die abstracts to be included in the model.
5. Similarly, create models for all fabrics up to the top layer by using references to the other die abstracts.

6. Choose *Simulation – Create and Edit* to run simulation and create n-ports that are saved in an S-param touchstone file.

Related Help Topic

Extracting Models for a Cross-Fabric Design

How to configure simulation settings?

You can configure the settings to be used by these simulators by using the Simulation Settings form. This form contains a specific tab for each simulator.

Related Help Topics

- Simulation Settings for Clarity
- Simulation Settings for EMX

How to create ports for models?

If the instances have pins or the selected nets have top-level pins, you can use the auto-generation feature to create ports that you can later review and adjust according to your specifications.

If you want full control on the placement of the ports and their edges, you can choose to create ports manually. Options for port configuration change depending on the simulator.

For more details, see:

How to create ports for Clarity models?

How to define a model for a full cellview?

If the IC layout is a passive device, we can characterize that layout cellview as an S-parameter cellview. In addition, you can create a symbol view, which you can use to instantiate the model in a schematic.

To specify that you need to extract a model for the full cellview:

- On the *Selection* tab in the Electromagnetic Solver assistant, select 

.

After the model is created, use the  (*Create S-Parameter View*) command on the toolbar of the assistant to create an s-parameter view and an optional symbol view for your cellview. The new views are added to the cell and are also displayed in the Library Manager. You can create instances of this newly-created s-parameter view in your design schematic.

Related Help Topic

Extracting S-Parameter Models from Complete Layout Cellviews

How to define a model for a layout-driven design?

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.

To create a model for a layout-driven flow:

1. Select *Extract Full Cellview* in the Electromagnetic Solver assistant.
2. Click *Edit the list of excluded cells* below the *Excluded Cells* list.
3. 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 active cell names, such as MOSFET, NFET, PFET, or BJT, that you need to exclude from the model.
5. (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.
6. Click *Automatically generate ports* on the Ports tab to generate ports for the cells in the model.
7. (Optional) Specify options on the *Pre-Process* tab to run shape simplification.
8. Generate and view mesh.

9. Run simulation for the model.
10. After the simulation, is complete, click *Create Simulation Schematic* to create a schematic using the S-parameters.
11. In the Create Simulation Schematic form, 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.

Related Help Topic

Extracting S-Parameter Models for a Layout-Driven Design

How to define a model for selected cells and instances?

When you are characterize selected parts of a layout, select those parts, nets or instances, for a model:

1. Select a model in the *Models* section of the Electromagnetic Solver assistant.
2. Select an instance or net on the layout canvas or in the Navigator assistant.
3. Click  on the *Selection* tab of the assistant.
   The selected shape is added to the *Object Selection* list.
4. Similarly, select more nets or instances.
5. If only a specific part of the selected nets or instances is relevant, create a cutting boundary around the selected objects.

   

6. When using the Clarity model, for an IC layout, you need to specify a few additional settings:
   a. *Current Return Path* on the *Selection* tab.
   b. The scale type and percentage for the die ground.

How to simplify shapes?

By simplifying shapes, you can reduce the 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. This improves the simulation runtime.

To simplify shapes:

1. Click *Setup* on the *Pre-Process* tab of the Electromagnetic Solver Assistant.
   The Simplify layout for EM simulation form is displayed.
2. Specify rules for the following purposes:
   a. To skip dummy metal fills from one or more layer-purpose pairs in the layout
   b. To remove dangling shapes, merge shapes, stripe shapes, or smooth shape steps
   b. To change the master for instances in the layout
   c. To remove dangling connected shapes from one or more layer-purpose pairs in the layout
   d. 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
3. (Optional) Click *Save As* to save the setup in a recipe file, which can be later loaded to reuse the settings.
4. Click *OK* to close the form.
5. Click *Preview Pre-Preprocessed Layout* on the *Pre-Process* tab.
   A preview of simplified shapes is displayed. If required, you can further modify the rules.
6. To protect certain shapes from modifications even if they meet the rules, select those shapes and click *Protect Selected* on the *Pre-Process* tab.

Related Help Topic

Simplifying Shapes

What are the steps involved in running a simulation?

The following topics describe the steps you need to perform to run electromagnetic simulations for different EM solvers:

- Running a simulation using Clarity

- Running a simulation using EMX

How to create an extracted view?

After the simulation is successfully complete, the status of the model is changed to `done` and the path to the S-parameters file saved with `.s2p` extension is shown in the *S-Params File* field on the *Results* tab.

To create an extracted view using the S-parameter file:

1. Click  (*Create Extracted View)* on the toolbar of the assistant.
   The Create Extracted View form is displayed. This form shows the library, cell, and view name for the extracted view. All available model files for the layout are listed in the *Model Files* list.
2. To allow the tool to automatically place the models in the schematic view, ensure that the *Interactive Mode* check box is deselected.
3. 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.
4. Click *OK*.

While saving 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. The model file becomes a part of the cellview. If you move or copy the cellview, the model file is moved or copied at the same time.

Related Content:

- Placing Models in the Schematic View Interactively
- Highlighting S-Parameter Models in Schematic

Highlighting Models in the Schematic View

After creating extracted an view, you can highlight the model in the schematic and review the S-parameter instances and nets.

1. Click  (Highlight S-Params in Schematic) on the toolbar of the Electromagnetic Solver assistant.
   The Backannotate from Extracted View form is displayed.
2. From the *Extracted View* list, select the name of the extracted view from which you want to highlight parameters in the schematic.
   The models and the N-port instances available in the extracted view are listed in the table on this form.
3. Select the instance to be highlighted on the schematic.
   The instances and wires for the nets included in the S-parameter file are highlighted in the schematic.

   The instances that are partially modeled are highlighted in yellow and those modeled completely are highlighted in red.

Placing Models in the Schematic View Interactively

To interactively place the models in an extracted view:

1. Click  (*Create Extracted View)* on the toolbar of the assistant.
   The Create Extracted View form is displayed. This form shows the library, cell, and view name for the extracted view. All available model files for the layout are listed in the *Model Files* list.
2. Select the *Interactive Mode* check box.
3. 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.
4. Click *OK*.
   The Place Components form is displayed. All the components for which models are available are displayed in the *All Components* list.
5. Select a mapped device in the *All Components* list.
6. Click *Place Selected Instances*.

The extracted view is displayed and the original instance of the selected device removed from it. Instead, an instance of model is highlighted in purple and attached to the mouse pointer.

7. Move the pointer and place the instance at the required location.
8. (Optional) Reconfigure the connections of the placed device model.
   After the model is successfully placed on the extracted view, the device name is moved to the *Connected Components* list on the Place Components form.
9. After all the devices are placed on the schematic view, close the Place Components form.