

# PCB RLC Extraction

China CPS Team

Xin Wang

# AppBrief Requirements(**Title matches the engineering challenge we are trying to solve (not simulation challenge)**)

## PCB RLC参数提取挑战

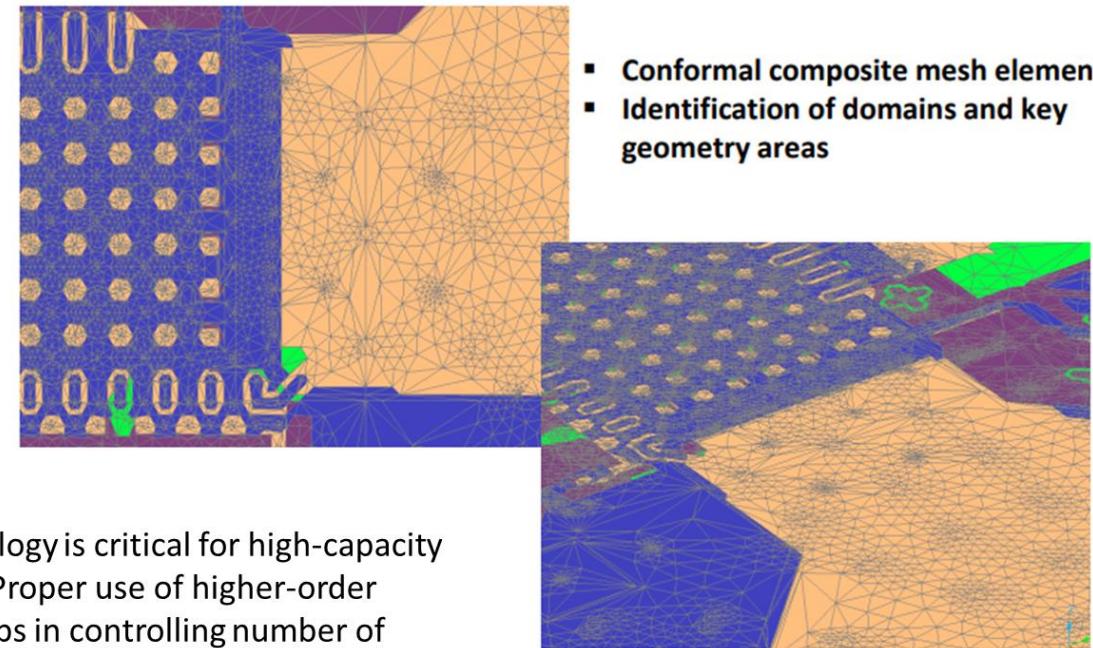
- PCB密度和集成度越来越高，传统的参数提取工具无法满足中型及超大PCB RLC提取的需求；
- 传统的参数提取工具对PCB叠层结构参数提取不准确且求解时间长；

## ANSYS解决方案

- ANSYS SIwave CPA采用**FEM Solver**，对信号线以及**source**和**sink**的数量没有限制，可以对提取成百上千**net**的RLC参数，和大规模的PDN结构；
- 包含**MoM Q3D Solver**，满足高精度参数提取的需求；

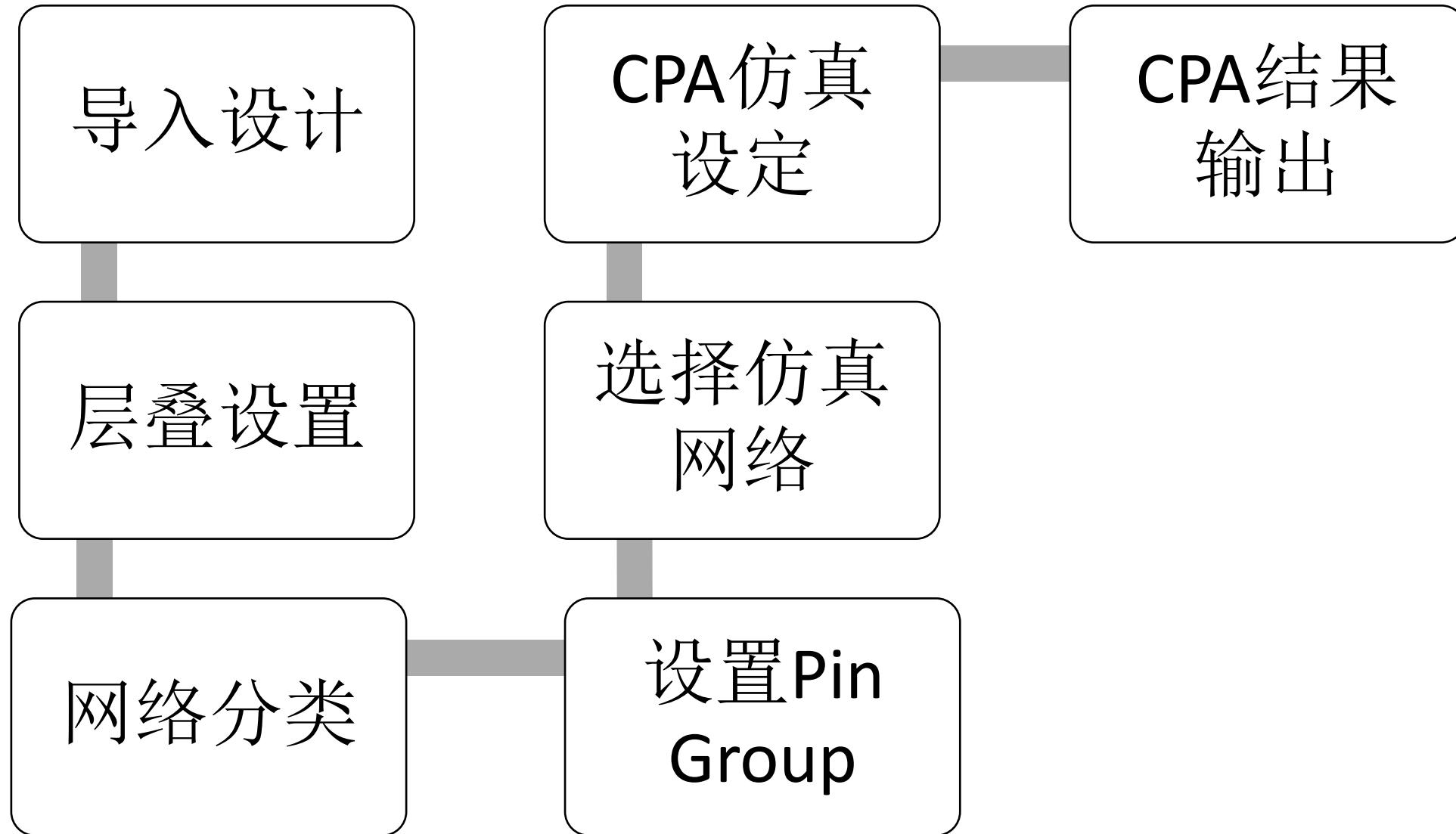
## Benefits

- 非常高容量的**FEM**求解器-全封装和PCB结构-上万个**source/sink**；
- 快速提取（几分钟到几小时-即使对与拥有上万个**net**的超大PCB结构）
- 适用多种结构：RDL、TSV、层压板PKG、PCB以及多Die等结构；

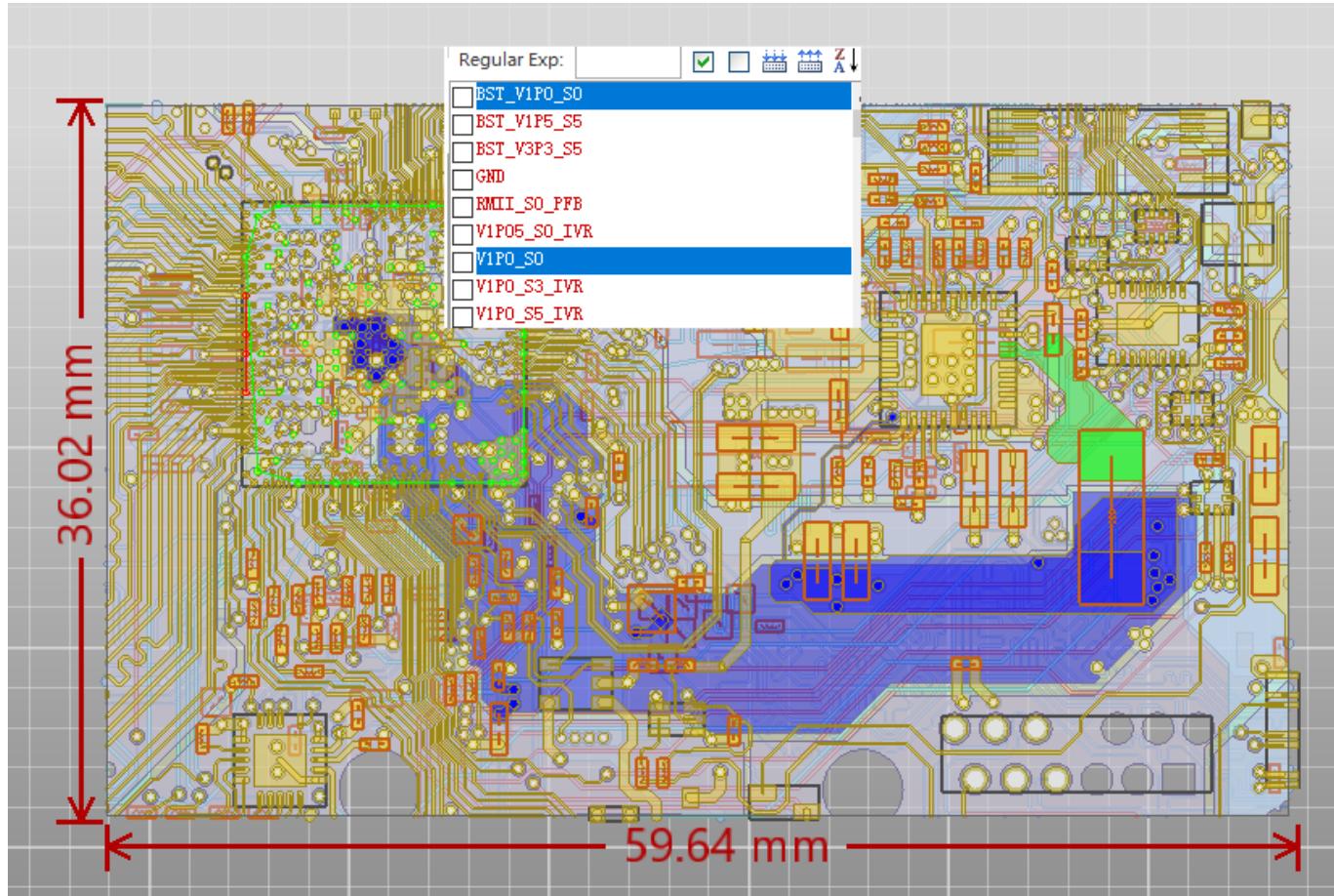


Mesh technology is critical for high-capacity simulations. Proper use of higher-order elements helps in controlling number of unknowns

# PCB RLC Extraction Workflow

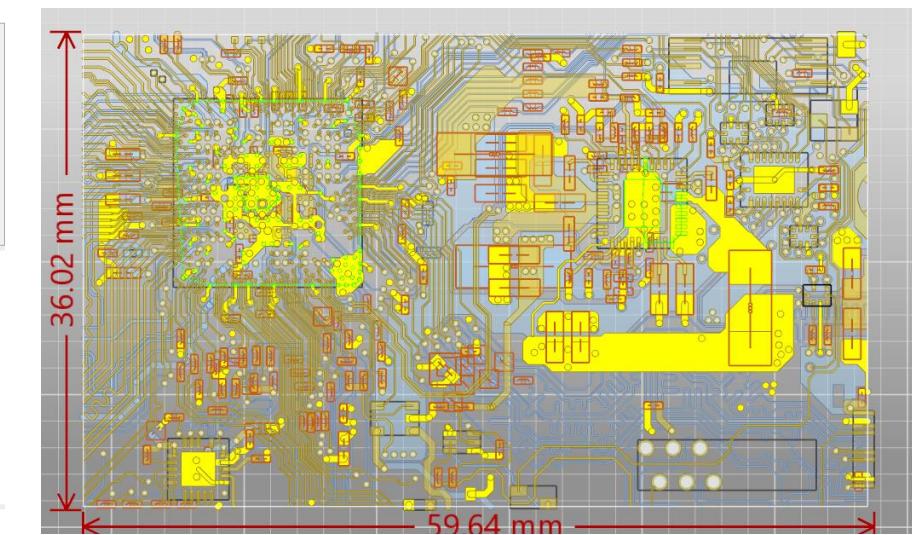
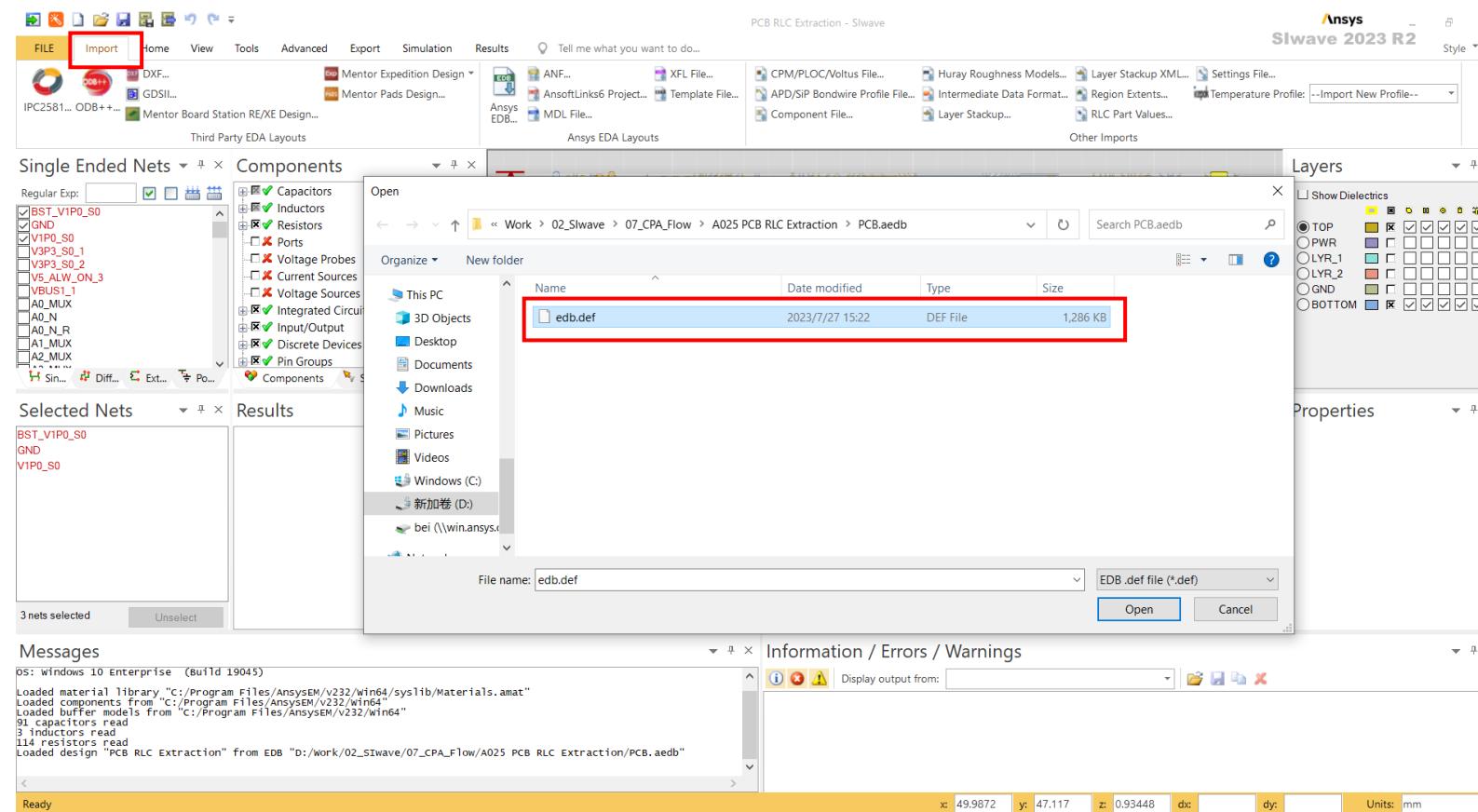


# 仿真demo背景



- 准备条件:
  - PCB设计文件
  - 层叠信息
- 仿真目标:
  - 提取电源网络BST\_V1P0\_S0、V1P0\_S0以及GND网络的寄生参数

# 导入设计



# 层叠设置

Siwave Workflow Wizard

- Import Component File...
- Import Stackup...
- Verify Stackup... (Selected)
- Verify Padstacks...
- Verify Circuit Element Parameters...
- Verify Power/Ground Net Classification...
- Sanitize Layout for Simulation
- Assign S-parameter Capacitor Mode...
- Assign SPICE Netlists...
- Configure DC IR Drop Analysis...
- Configure PI Analysis...
- Configure DDR Analysis...

Always show this dialog after project import

Layer Stackup Editor

C...	Name	Type	Thickness (mm)	Material	Conductivity (S/m)	Dielectric const.	Dielectric con...	Loss tan...	Translucen...	Elevation (mm)	Roughness (m...)	Trace Cross-s...
	DIELECTRI...	CONFOR...	0.015	SolderMask	0	3.1	0.035	0.035	0.98274			
	TOP	METAL	0.04826	EDB_COPPER	5.959E+07	SolderMask	3.1	0.035	60	0.93448	HJ: 0 , HJ: 0 , HJ: 0	Rectangle
	UNNAMED...	DIELECTR...	0.06731	FR-4	0	4.5	0.035	0.035	0.86717			
	PWR	METAL	0.03302	EDB_COPPE...	5.959E+07	FR-4	4.5	0.035	60	0.83415	HJ: 0 , HJ: 0 , HJ: 0	Rectangle
	UNNAMED...	DIELECTR...	0.2032	FR-4	0	4.5	0.035	0.035	0.63095			
	LYR_1	METAL	0.03048	EDB_COPPE...	5.959E+07	FR-4	4.5	0.035	60	0.60047	HJ: 0 , HJ: 0 , HJ: 0	Rectangle
	UNNAMED...	DIELECTR...	0.2032	FR-4	0	4.5	0.035	0.035	0.39727			
	LYR_2	METAL	0.03048	EDB_COPPE...	5.959E+07	FR-4	4.5	0.035	60	0.36679	HJ: 0 , HJ: 0 , HJ: 0	Rectangle
	UNNAMED...	DIELECTR...	0.2032	FR-4	0	4.5	0.035	0.035	0.16359			
	GND	METAL	0.03302	EDB_COPPE...	5.959E+07	FR-4	4.5	0.035	60	0.13057	HJ: 0 , HJ: 0 , HJ: 0	Rectangle
	UNNAMED...	DIELECTR...	0.06731	FR-4	0	4.5	0.035	0.035	0.06326			
	BOTTOM	METAL	0.04826	EDB_COPPER	5.959E+07	SolderMask	3.1	0.035	60	0.015	HJ: 0 , HJ: 0 , HJ: 0	Rectangle
	DIELECTRI...	CONFOR...	0.015	SolderMask	0	3.1	0.035	0.035	0			

Add / Delete / Move Layer(s)      Edit Selected Layer(s)

Add Above Selected Layer      Color: c4ab1e      Update  
Add Below Selected Layer      Name: TOP      Update  
Delete Selected Layers      Translucency: 60%      Update  
Move Selected Layers Up      Type: METAL      Thickness: 0.04826 mm      Update  
Move Selected Layers Down      Material: EDB\_COPPER      Roughness: HJ: 0 , HJ: 0 , HJ: 0 mm      Update

Select all DIELECTRIC layers      Scheme-1      Save Current Color Scheme      Color scheme As Is      Set as Default      Default Scheme for new projects: Siwave

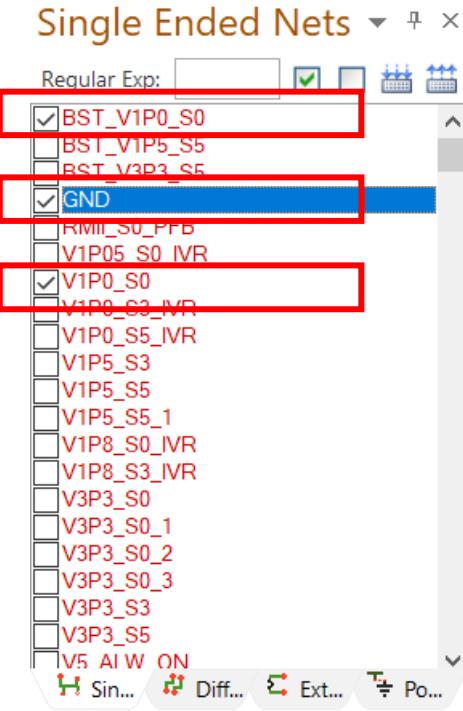
Thickness change affects die elevations      Edit Material Properties      Invert Stackup       Conformal Coat      Units: mm       Copper Weight      OK      Cancel

# 网络分类



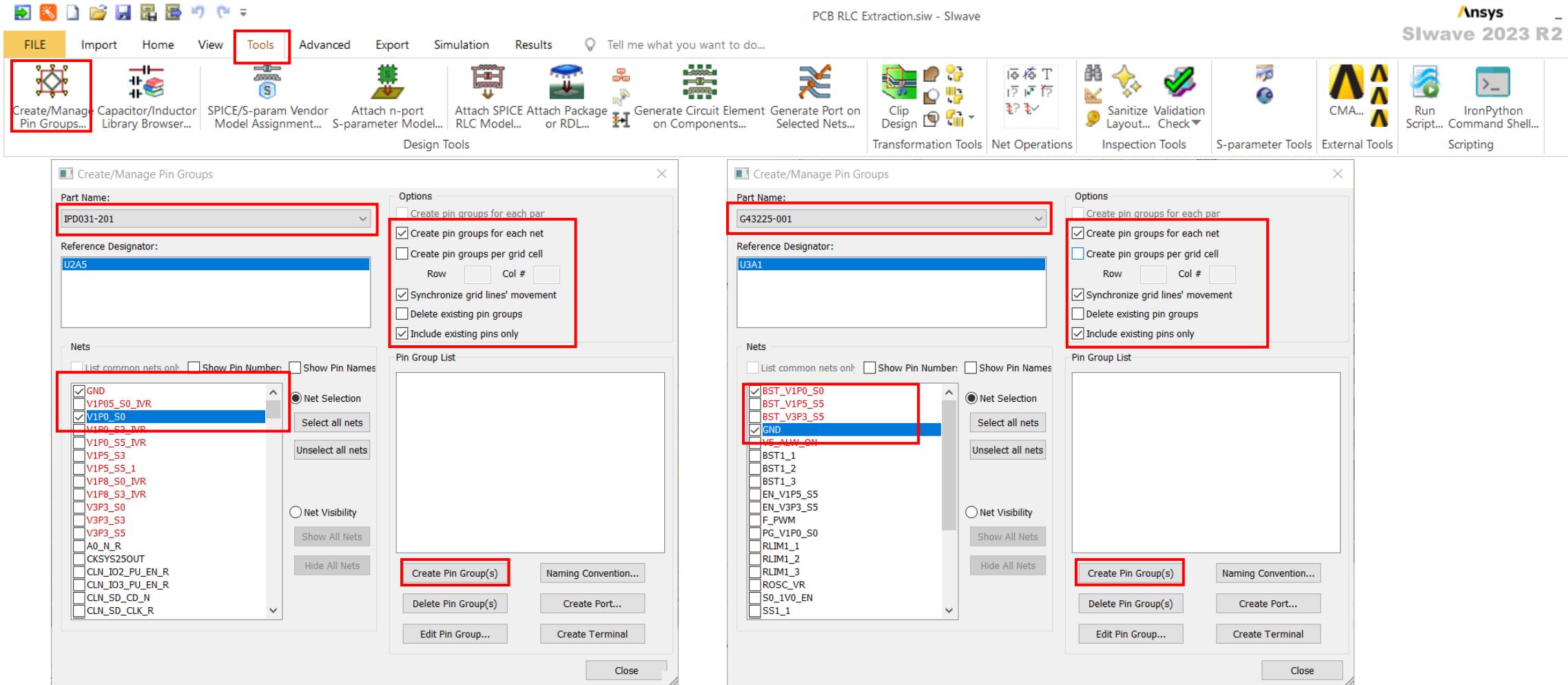
对电源和地网络进行分类

# 选择要仿真的net



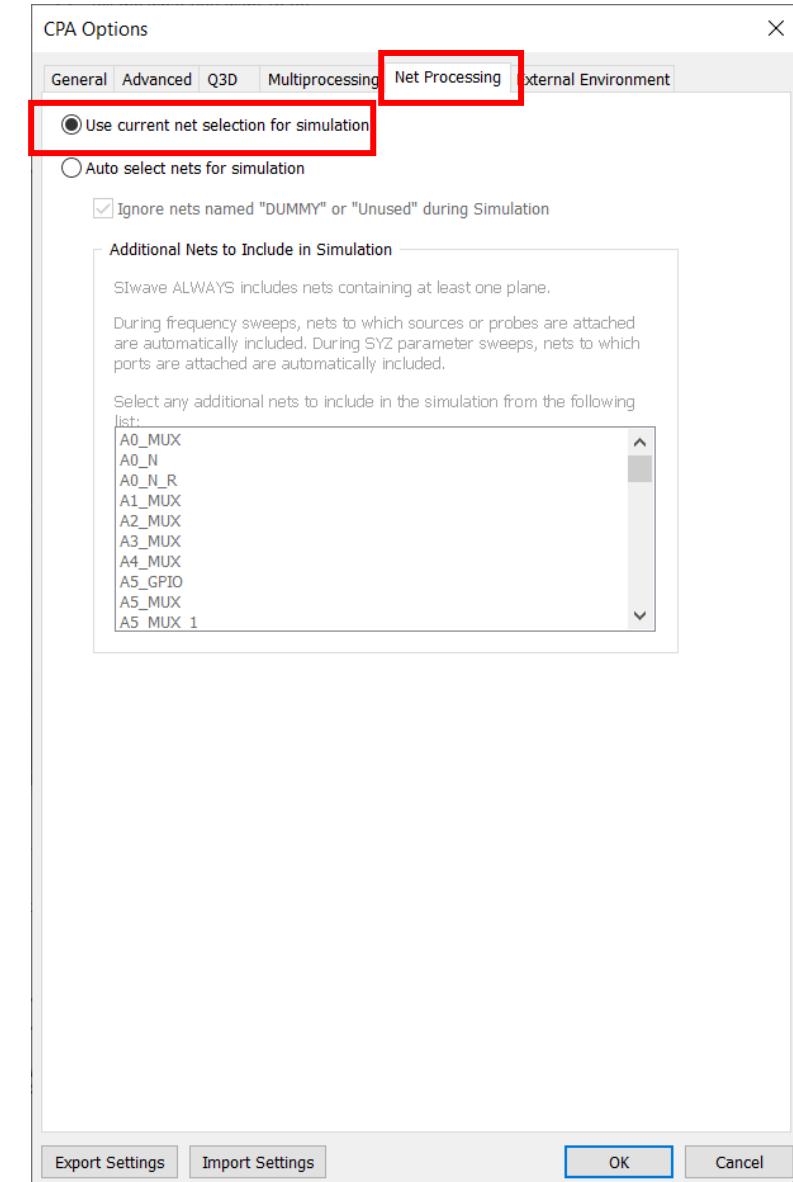
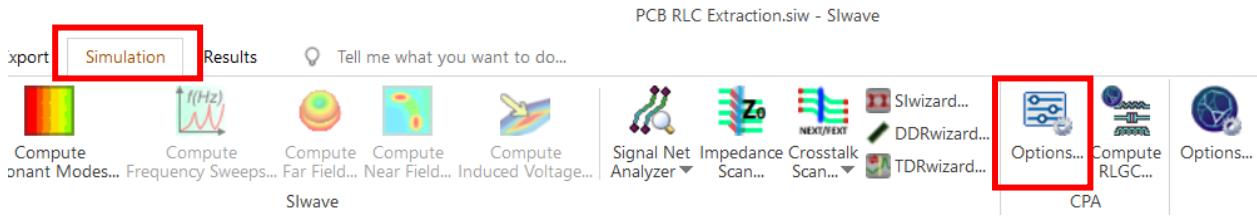
选择电源网络BST\_V1P0\_S0、V1P0\_S0以及GND网络

# 设置 Pin Group

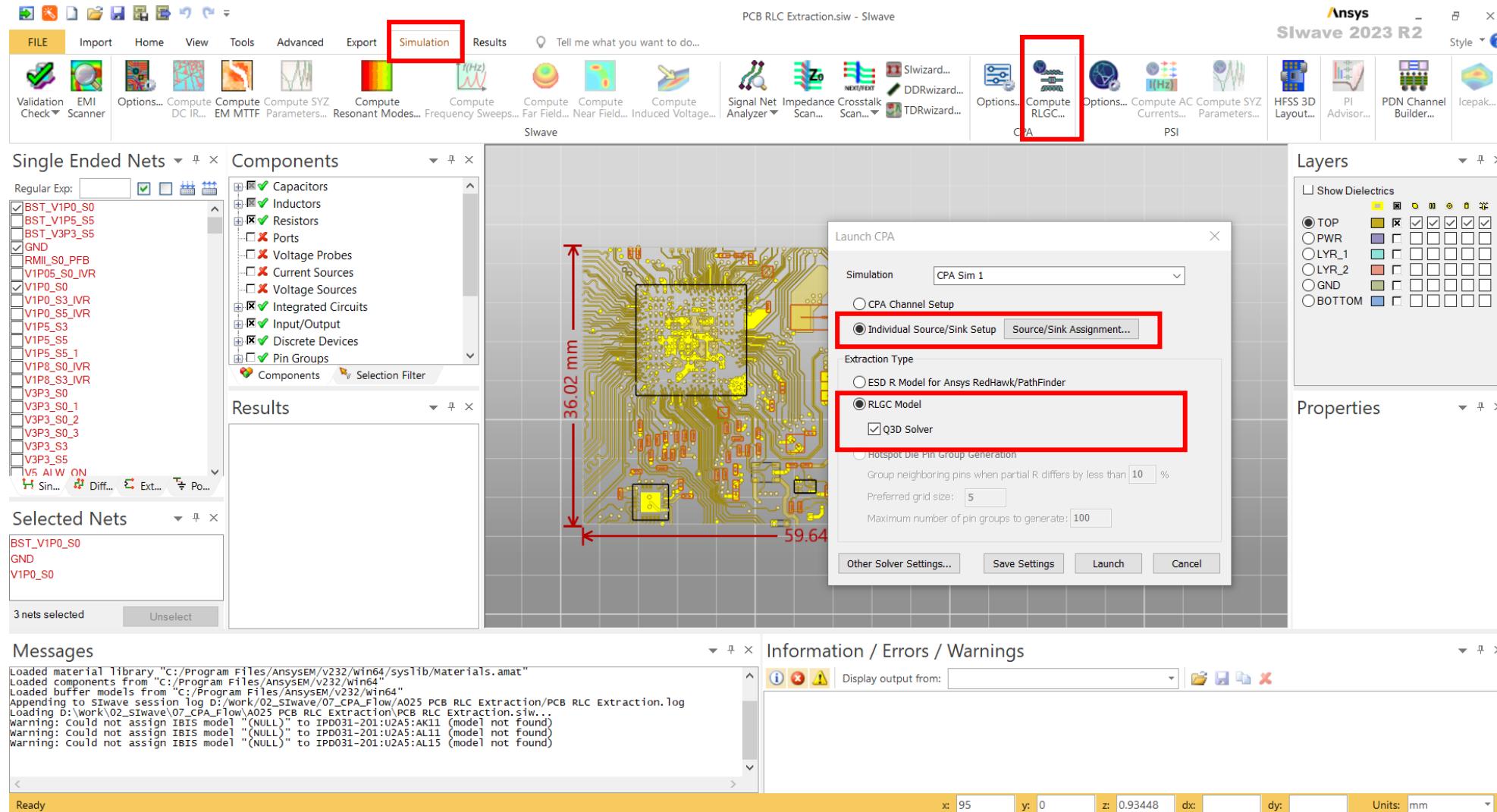


选择对应器件分别对要仿真的Nets进行Pin Group

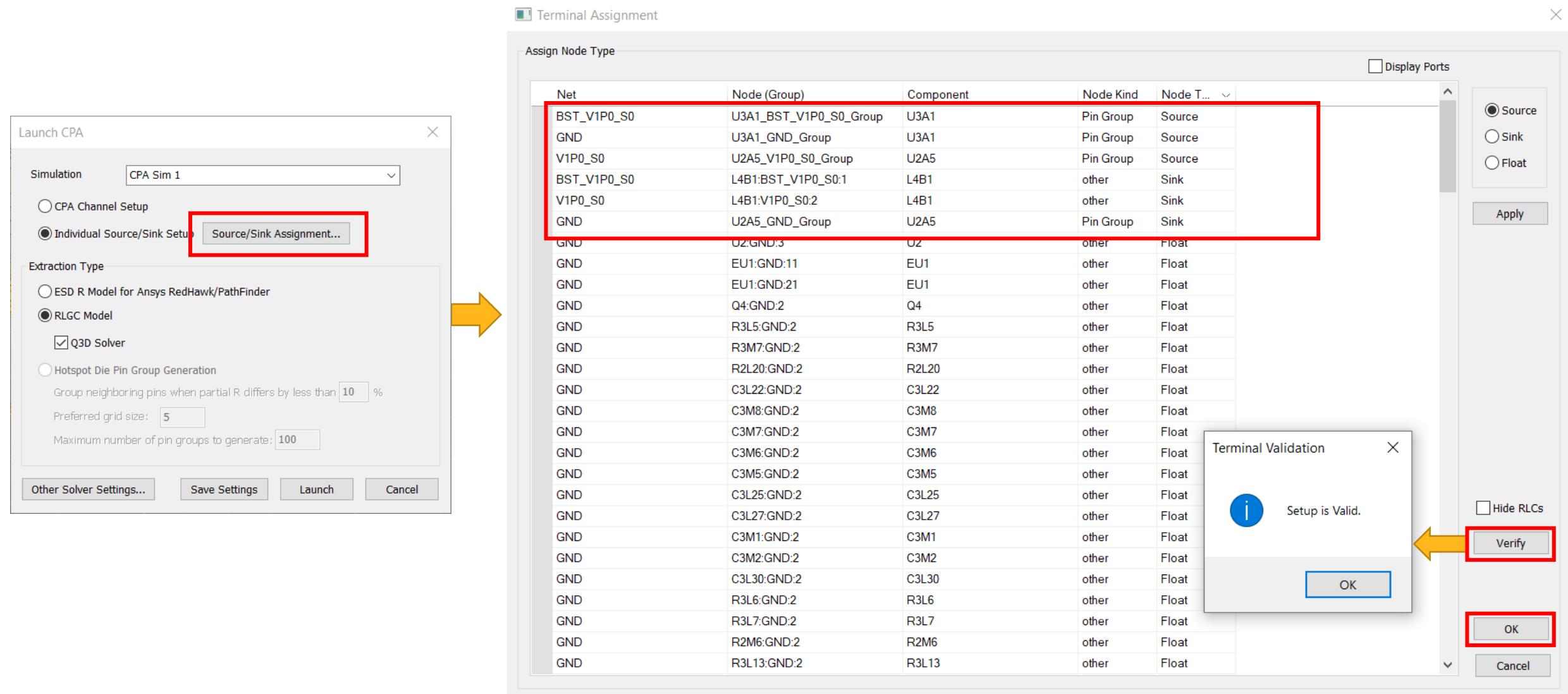
# CPA Option



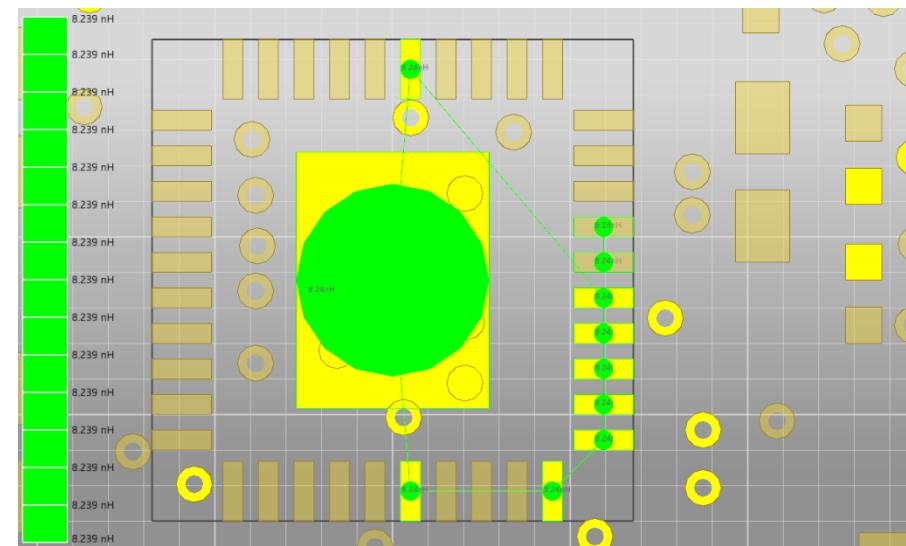
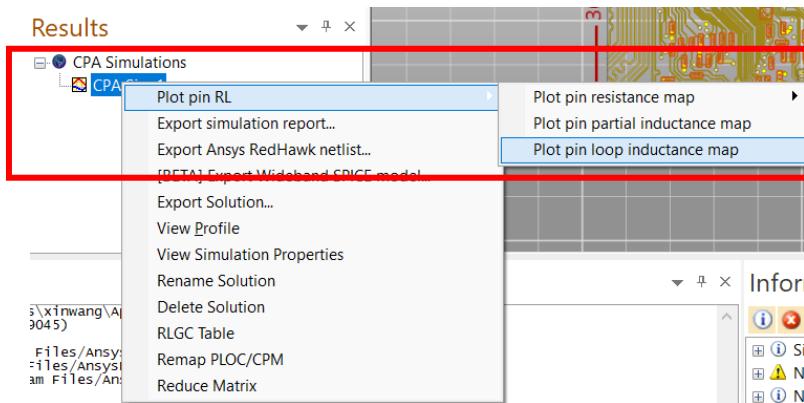
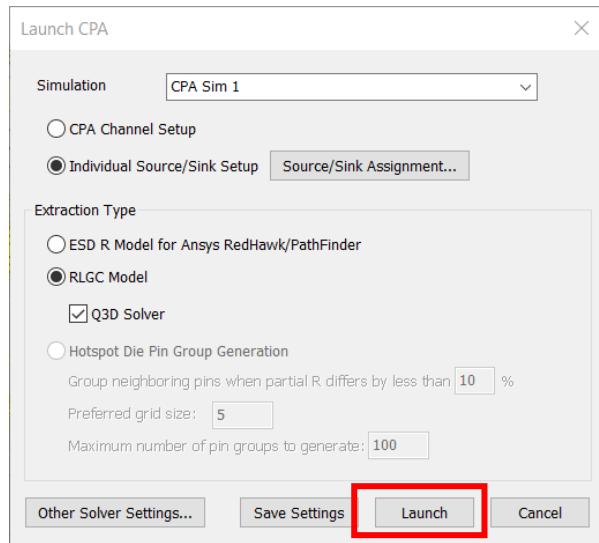
# 方法1：选择individual模式：采用Q3D求解器



# 方法1：选择individual模式：设置source、sink



# 方法1：选择individual模式：采用Q3D求解器

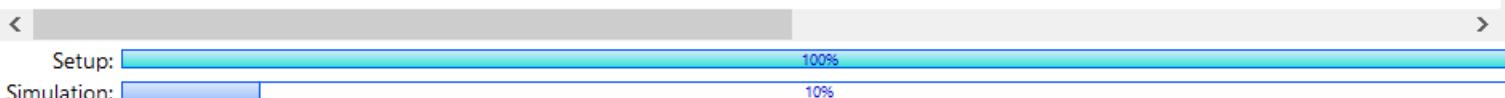


Process Monitor (CPA Sim 1)

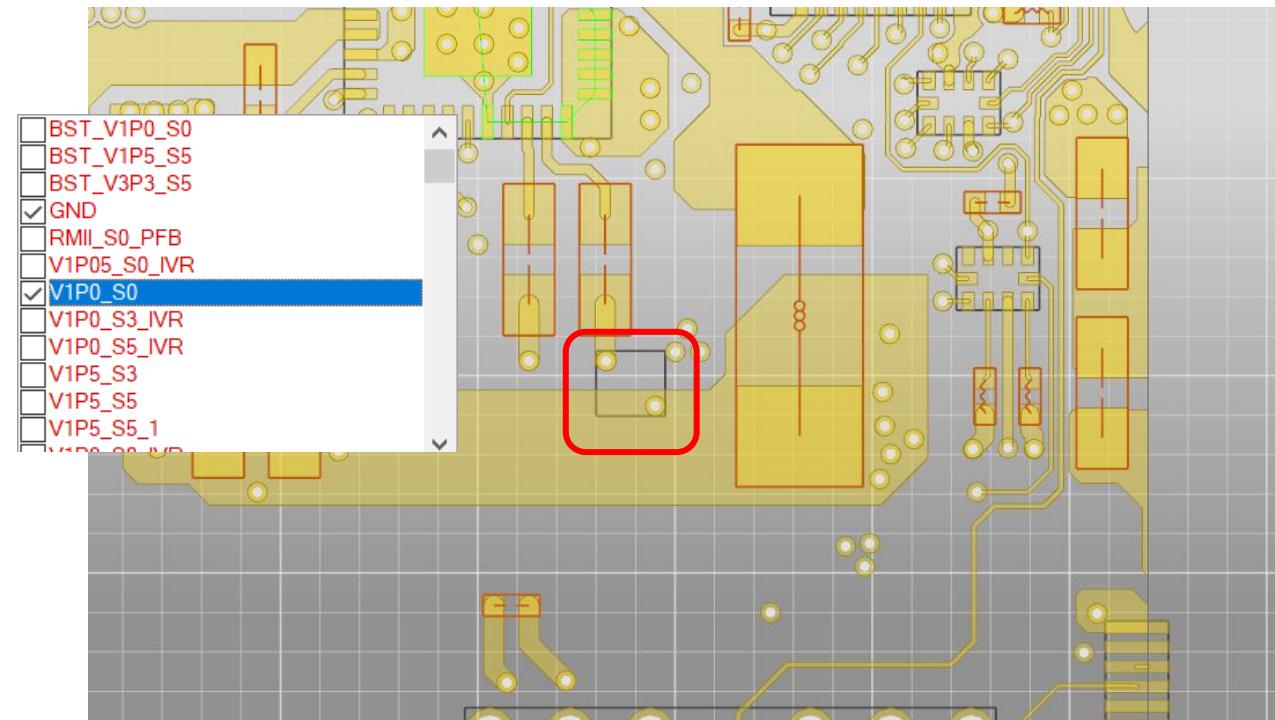
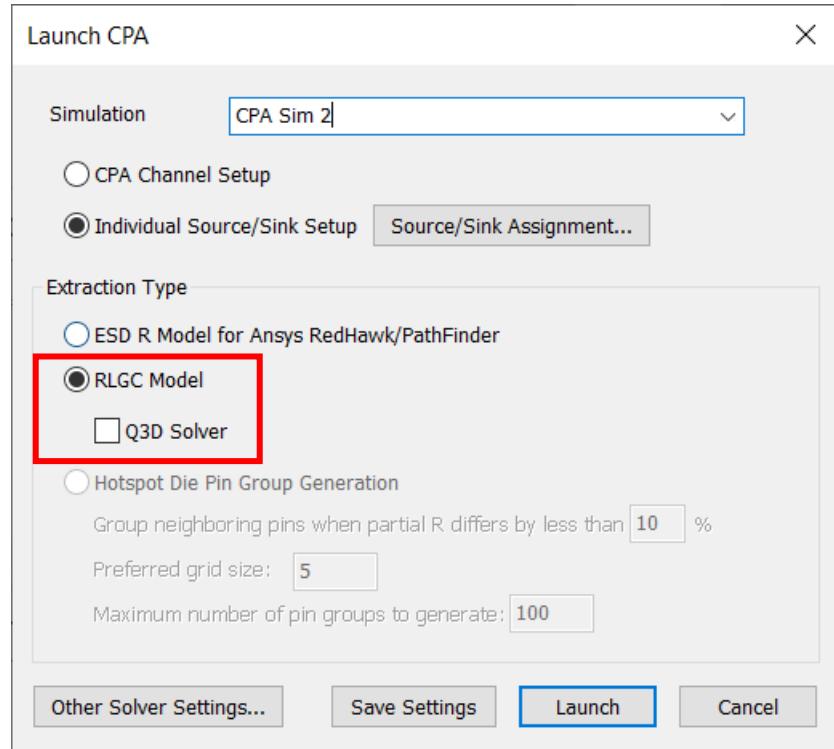
```
Display: Messages ▾ ■ ▶
E16347-001_C3B17 :: E16347-001
***** EOF Components in the simulation *****

Number of cores = 4

Package EM extraction in progress ...
* CPA-Q3D CG Extraction in progress
CG solve Maximum adaptive passes = 4 Target percent error = 5.000
Start CG solution in beixwang
```



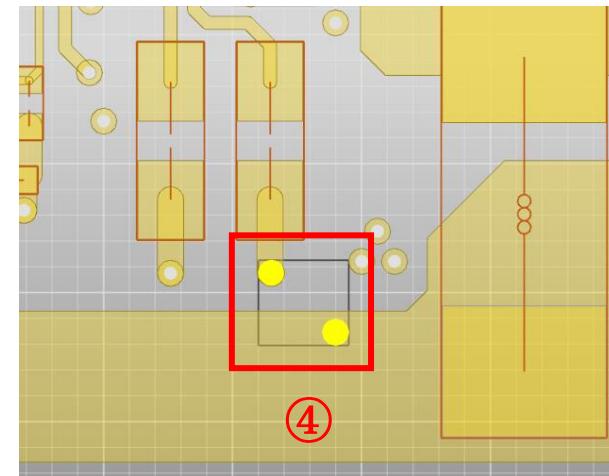
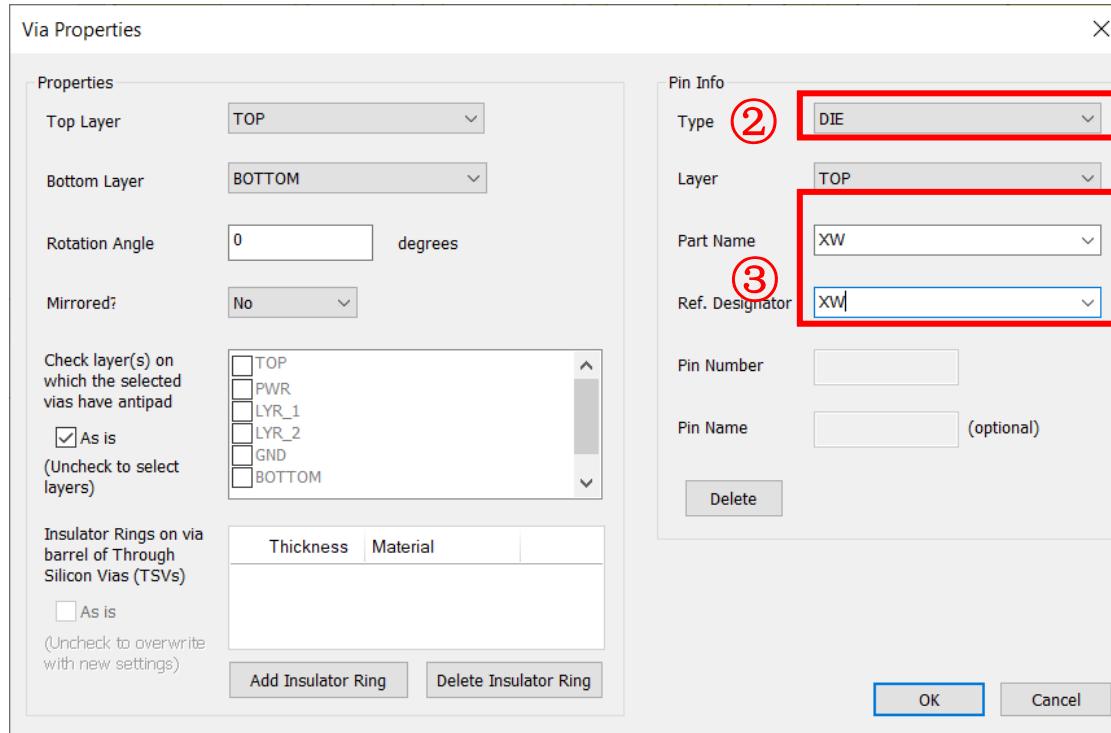
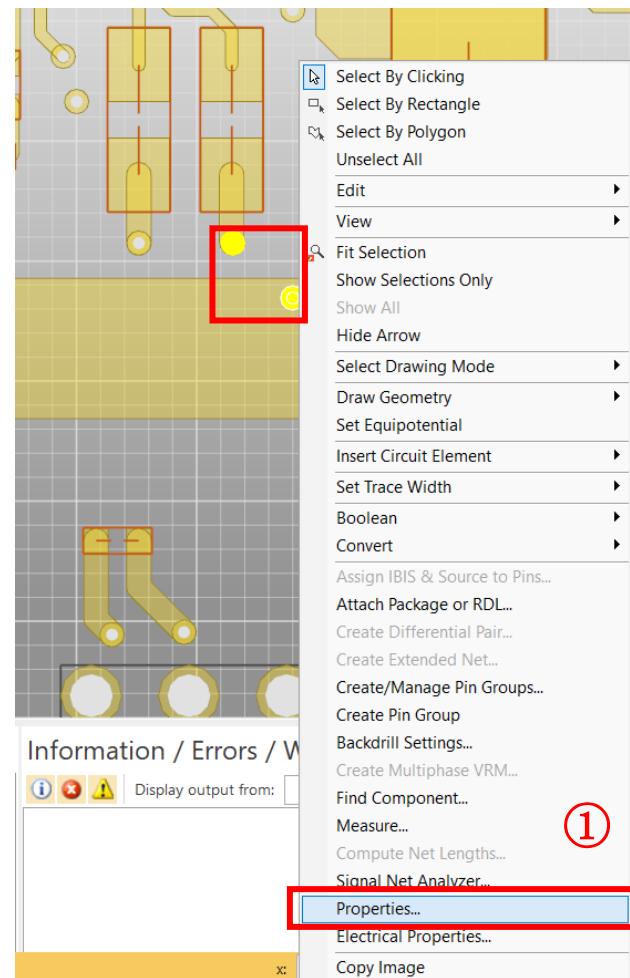
# 方法2：选择individual模式：采用CPA求解器



以提取V1P0的网络为例，也就是需要在电感下端和附件的GND创建一个假器件

注意：CPA FEM求解器需要计算回路参数，由于中间电感处没有对应的GND引脚，所以没有办法直接运行仿真。软件要求每个信号网络对应的GND网络应该在网络的同一端存在一个器件中，所以可通过添加假器件的方法来解决，分布仿真的方法解决。

# 方法2：选择individual模式： 创建假器件方法



选中要创建假器件的过孔（需要给电源和GND设置source、sink的位置）打开属性框，编辑type和名称，点击OK

# 方法2：选择individual模式：设置source、sink

The screenshot illustrates the process of setting up individual source/sink assignments for a component. It shows two windows: the 'Launch CPA' dialog and the 'Terminal Assignment' configuration window.

**Launch CPA Dialog:**

- Simulation: CPA Sim 2
- Extraction Type:
  - RLGC Model (selected)
  - Q3D Solver (unchecked)
- Other Solver Settings...
- Save Settings
- Launch
- Close

**Terminal Assignment Window:**

The 'Assign Node Type' table lists terminal assignments:

Net	Node (Group)	Component	Node Kind	Node T...
V1P0_S0	U2A5_V1P0_S0_Group	U2A5	Pin Group	Source
GND	U2A5_GND_Group	U2A5	Pin Group	Source
V1P0_S0	XW:V1P0_S0:2	XW	other	Sink
GND	XW:GND:1	XW	other	Sink
GND	U2:GND:3	U2	other	Float
GND	EU1:GND:11	EU1	other	Float
GND	EU1:GND:21	EU1	other	Float
GND	Q4:GND:2	Q4	other	Float
GND	R3L5:GND:2	R3L5	other	Float
GND	R3M7:GND:2	R3M7	other	Float
GND	R2L20:GND:2	R2L20	other	Float
GND	C3L22:GND:2	C3L22	other	Float
GND	C3M8:GND:2	C3M8	other	Float
GND	C3M7:GND:2	C3M7	other	Float
GND	C3M6:GND:2	C3M6	other	Float
GND	C3M5:GND:2	C3M5	other	Float
GND	C3L25:GND:2	C3L25	other	Float
GND	C3L27:GND:2	C3L27	other	Float
GND	C3M1:GND:2	C3M1	other	Float
GND	C3M2:GND:2	C3M2	other	Float
GND	C3L30:GND:2	C3L30	other	Float
GND	R3L6:GND:2	R3L6	other	Float
GND	R3L7:GND:2	R3L7	other	Float
GND	R2M6:GND:2	R2M6	other	Float
GND	R3L13:GND:2	R3L13	other	Float
GND	R3L4:GND:2	R3L4	other	Float
GND	C3L21:GND:2	C3L21	other	Float

A red box highlights the first four rows of the table, corresponding to the 'RLGC Model' setup in the Launch CPA dialog.

**Terminal Validation Dialog:**

Setup is Valid.

OK Cancel

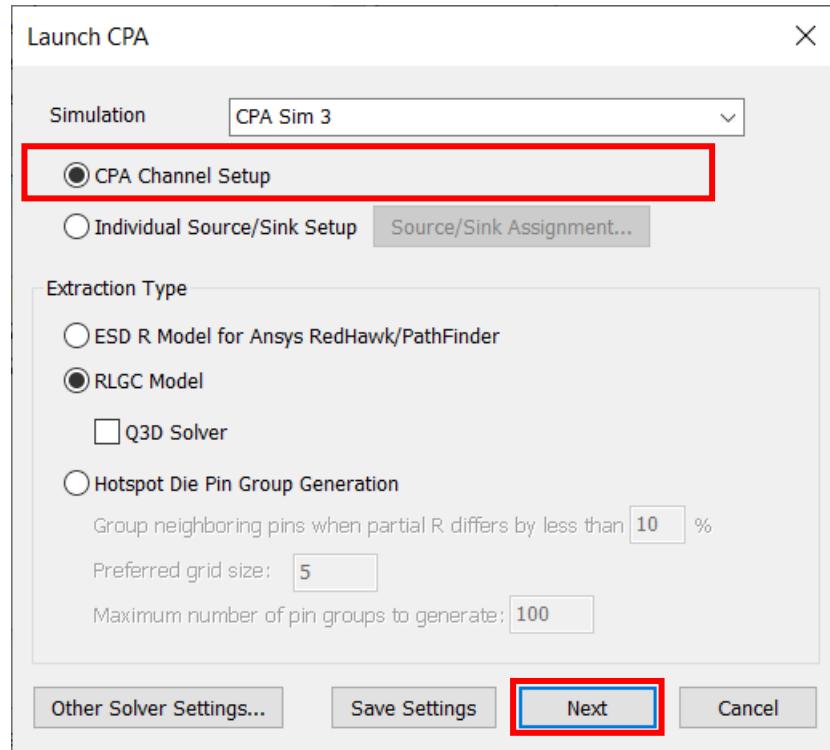
Verify Hide RLCs

# 方法2：选择individual模式：采用CPA求解器

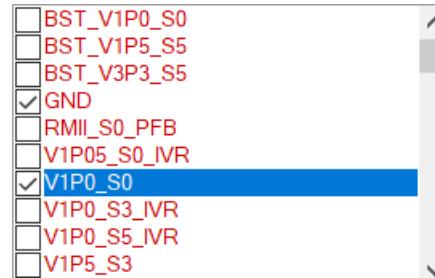
The screenshot displays three main windows related to the CPA求解器 (CPA Solver) process:

- Launch CPA** window (left): Shows the "Simulation" dropdown set to "CPA Sim 2". The "Individual Source/Sink Setup" option is selected. Under "Extraction Type", the "RLGC Model" is chosen. Other settings include "Preferred grid size: 5" and "Maximum number of pin groups to generate: 100". Buttons for "Save Settings", "Launch" (highlighted with a red box), and "Close" are at the bottom.
- Results** window (center): A tree view shows "CPA Simulations" with "CPA Sim 1" and "CPA Sim 2" expanded. A context menu is open over "CPA Sim 2", with options like "Plot pin RL", "Plot pin resistance map", "Plot pin partial inductance map", and "Plot pin loop inductance map" highlighted with a red box. The background shows a circuit board layout with various components and pins.
- Process Monitor (CPA Sim 2)** window (bottom): Displays log messages:
  - \* Ref Net = GND
  - \* Global Die = IPD031-201\_U2A5
  - \* Global VRM = XW\_XW
  - \* Pin groups defined on XW\_XW are used for extraction
  - \* Net = V1P0\_S0 sink Comp = XW\_XWA progress bar at the bottom indicates "Setup: 100%" and "Simulation: 10%".

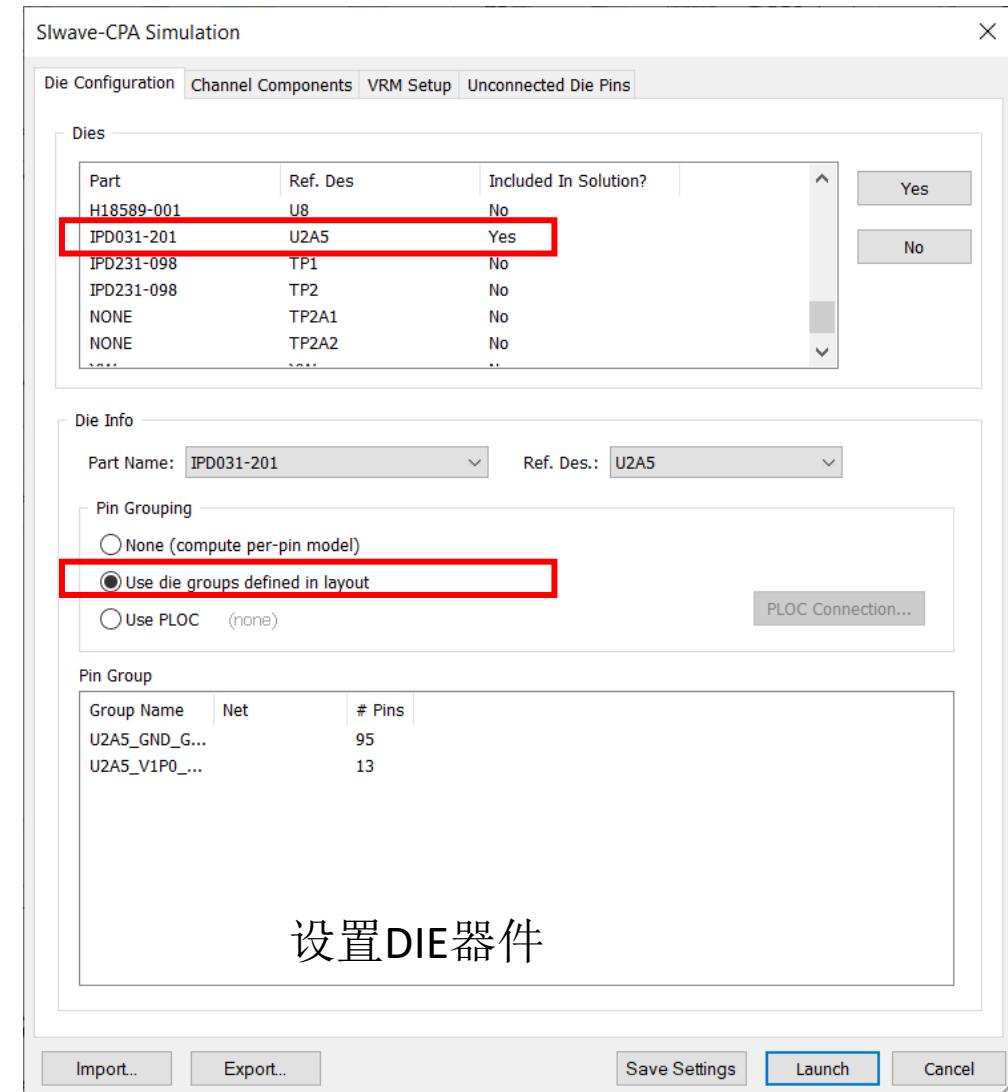
# 方法3：选择CPA Channel模式：采用CPA求解器



采用CPA channel模式

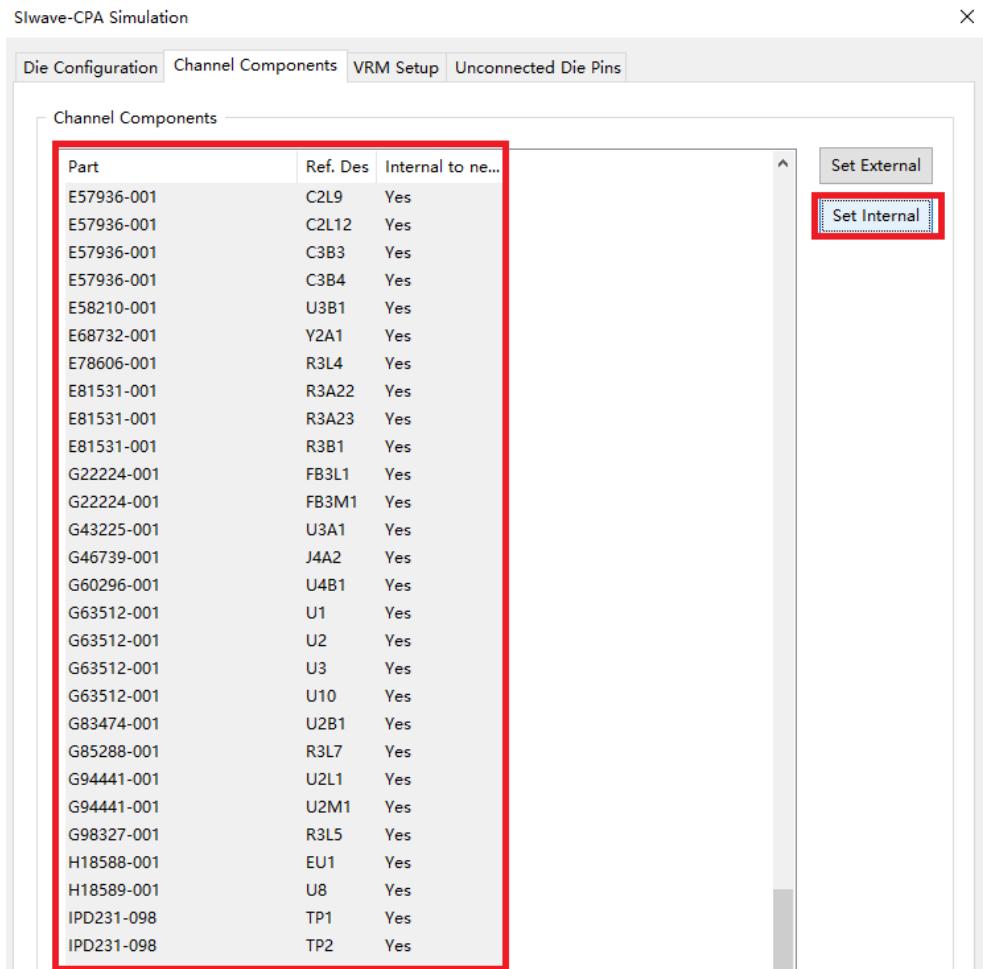


选择仿真网络

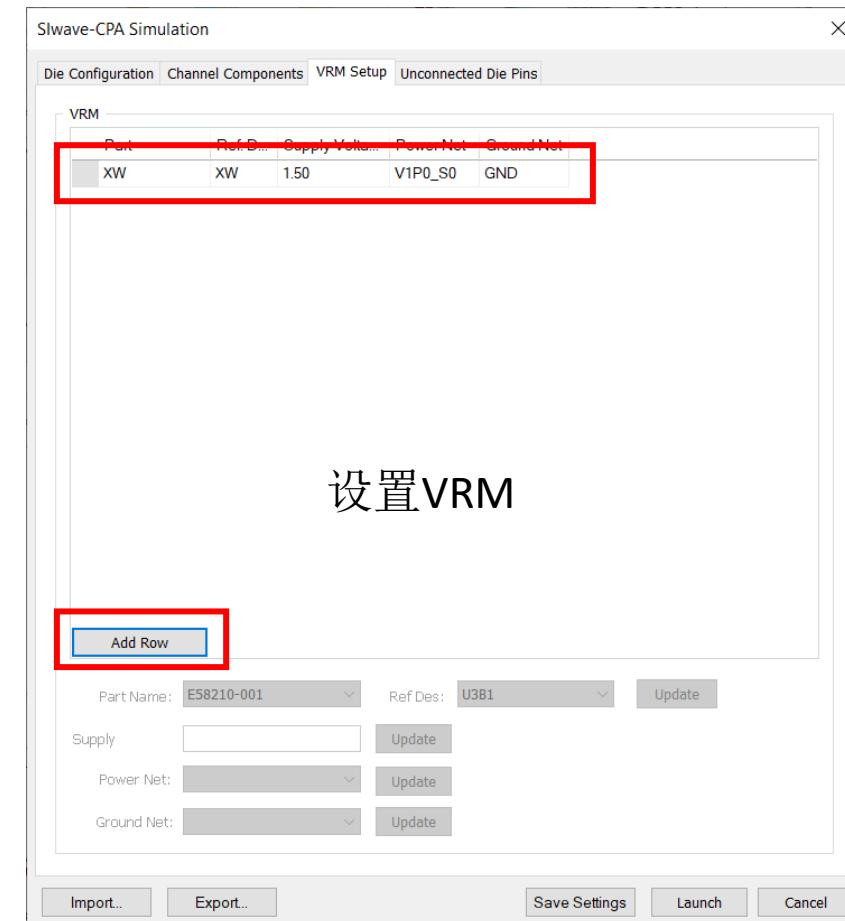


设置DIE器件

# 方法3：选择CPA Channel模式：采用CPA求解器



将网络上的元件当成内部元件处理



设置VRM

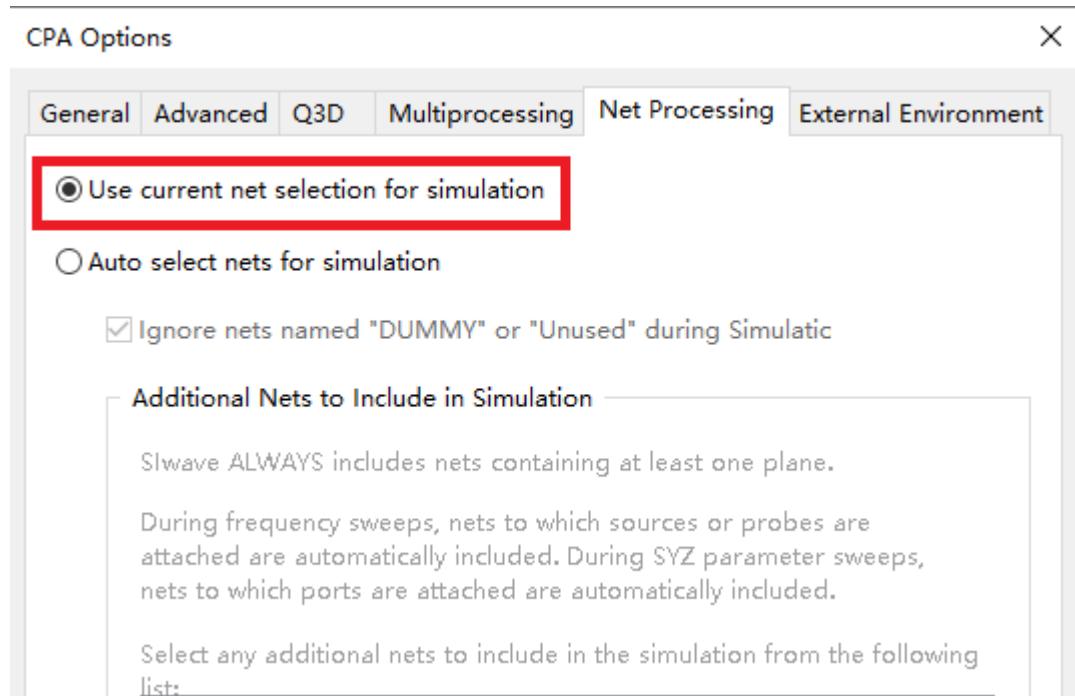
VRM的设定中，至少或只需要设置一个器件，器件位于仿真的某个网络和GND之间即可。

# 方法3：选择CPA Channel模式：采用CPA求解器

The screenshot displays the Ansys Siwave-CPA Simulation interface across three main windows:

- Siwave-CPA Simulation Window:** Shows the VRM setup configuration. A red box highlights the "Launch" button at the bottom center of the dialog.
- Results Window:** Displays simulation results for CPA Sim 3. A red box highlights the context menu for CPA Sim 3, which includes options like "Plot pin RL", "Plot pin resistance map", and "Plot pin loop inductance map". To the right is a heatmap showing pin loop inductance values ranging from 0.736 nH (blue) to 2.814 nH (red).
- Process Monitor (CPA Sim 3) Window:** Shows the command-line interface for the simulation, displaying component assignments and EOF messages. It also includes progress bars for Setup (100%) and Simulation (10%).

# 注意事项



1、CPA设置中设置如上图，对目前选择的网路进行仿真

- 2、不采用Q3D求解的时候，必须选择两个以上网络  
软件需要计算回路参数，需要有网络当成回路。
- 3、仿真前要对设计进行检查，不能有短断路的情况
- 4、电源和参考GND需要在网络的两端都有器件连接

# **Appendix**

## **ALH注册 CPS中文学习室**



# CPS中文学习室

**适用人群:** 先进封装和系统的SIPI、Thermal、Mechanical工程师。

**设计目标:** 让工程师快速获得最有用、最必要的相关知识（文档、录像、教程）。

## 主要架构

- **入门必读:** 包含软件基本功能。每个工程师全部下载并通学。
- **高级技巧:** 包含最新的高阶流程。需要的工程师专题学习。
- **标准教程:** ALH自带标准软件教程。学习软件操作时使用。

## 进入方式:

- **已有ALH帐号:** 点击以下链接：

[https://jam8.sapjam.com/groups/oOWAdwLABmWAIBypw\\_wDR5uv/overview\\_page/SJgVYUCTz90LbjxUQLHMSH](https://jam8.sapjam.com/groups/oOWAdwLABmWAIBypw_wDR5uv/overview_page/SJgVYUCTz90LbjxUQLHMSH)

- **没有ALH帐号:** 参考第二页注册ALH账号后进入。

- **联系人:** [xiaoxia.zhou@ansys.com](mailto:xiaoxia.zhou@ansys.com)

The screenshot shows the homepage of the CPS Chinese Learning Room. At the top, there's a navigation bar with tabs for CPS, SIPI, Thermal, and Mechanical. Below the navigation is a search bar and a user profile icon. The main content area features a banner for the 'SIPI Learning Room' with various circuit board and connector diagrams. To the right of the banner is a section titled 'Main Features' with three items: 'Precision Priority', 'Efficiency Improvement', and 'Multi-physics Field'. Below this are two sections: '① Basic Skills' and '② Advanced Techniques', each listing several PDF and procedure documents. A 'Latest Version' section indicates '2023R1.1 Download'. At the bottom right is a 'New Feature Highlight' section with links to various features like 'Phi+ Mesh Fusion' and 'Single AMI Model Simulation'.

# ALH注册方法

打开下面网址或扫描右边二维码进行注册：

[https://www.ansys.com/alhactivation\\_cn](https://www.ansys.com/alhactivation_cn)

第13步“是否有激活码”，请选择“否”

第14步“请输入您的激活码”，请填写客户号。

客户号为一串6位或7位数字。

获取方法：

- 询问Ansys客户经理/技术。
- 启动软件，观察Help – About - Client License Settings - Customer Number。



Ansys

