

© 2026 Synopsys, Inc and/or ANSYS, Inc. All rights reserved.
Unauthorized use, distribution, or duplication is prohibited.

Getting Started with Icepak®: Toolkits

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
https://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2026 R1
January 2026

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

Copyright and Trademark Information

© 1986-2026 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for Ansys proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Conventions Used in this Guide

Please take a moment to review how instructions and other useful information are presented in this documentation.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
  - Keyboard entries that should be typed in their entirety exactly as shown. For example, "**copy file1**" means you must type the word **copy**, then type a space, and then type **file1**.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). For example, "click **HFSS** > **Excitations** > **Assign** > **Wave Port**."
  - Labeled keys on the computer keyboard. For example, "Press **Enter**" means to press the key labeled **Enter**.
- Italic type is used for the following:
  - Emphasis.
  - The titles of publications.
  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, "**copy** *filename*" means you must type the word **copy**, then type a space, and then type the name of the file.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, "Press Shift+F1" means to press the **Shift** key and, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Draw** > **Line**"



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This instruction means that you should click the **Line** command on the **Draw** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *shortcut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation> Wave Port**” means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the shortcut menu (and the corresponding sub-menus).

Getting Help: Ansys Technical Support

For information about Ansys Technical Support, go to the Ansys corporate Support website, http://www.ansys.com/Support. You can also contact your Ansys account manager in order to obtain this information.

All Ansys software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Help Menu

To access help from the Help menu, click **Help** and select from the menu:

- **[product name] Help** - opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** - opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** - opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table of Contents

Table of Contents **Contents-1**

1 - Introduction **1-1**

Set 3D UI Options 1-1

2 - Set Up the Project **2-1**

Launch the Ansys Electronics Desktop 2-1

Create a PBGA Package 2-1

3 - Analyze and Extract Theta JB **3-1**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 - Introduction

Toolkits provide means to automate routine tasks and help maximize efficiency of project workflows. The Ansys Electronics Desktop offers an array of Icepak toolkits organized by geometry creation, modeling, and productivity. This document is intended as supplementary material to Icepak for beginners and advanced users. It includes instructions to use the Icepak Create PGBA toolkit to create the model and Extract JB and JC toolkit to solve the model and extract the theta JB value.

Set 3D UI Options

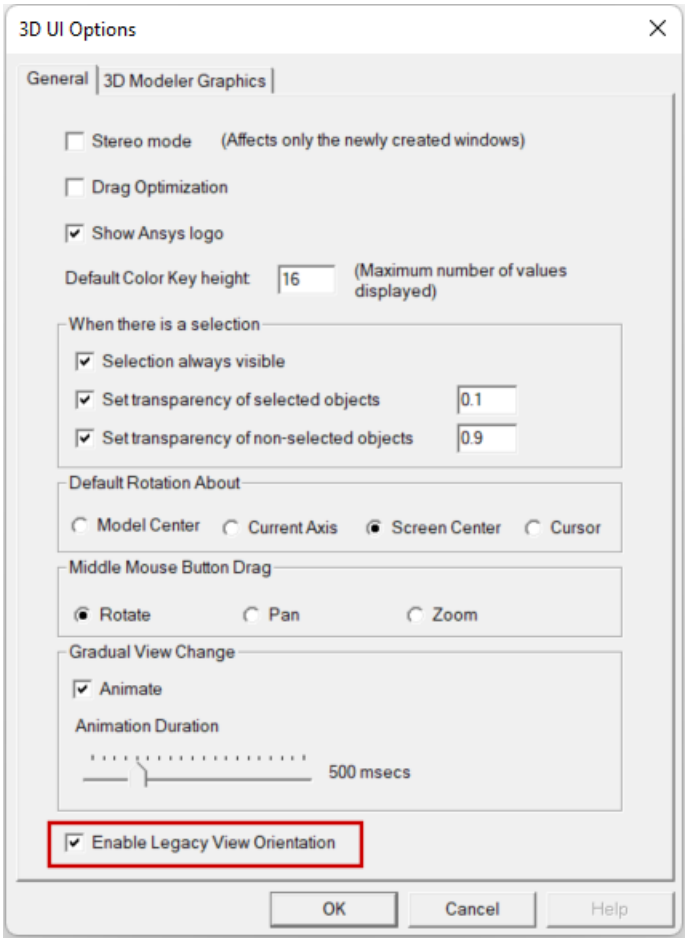
Ensure that the new view orientation scheme introduced in release 2024 R1 is not being used, since the instructions and images in this guide are based on the legacy orientation scheme.

1. From the menu bar, click **View** > **Options**.

   The *3D UI Options* dialog box appears.

2. Ensure that **Enable Legacy View Orientation** is enabled:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 - Set Up the Project

Launch the Ansys Electronics Desktop

A shortcut of the Ansys Electronics Desktop application appears on your desktop once the application is installed.

1. On the **Desktop** ribbon tab, click **Icepak** to insert an Icepak design in the project.

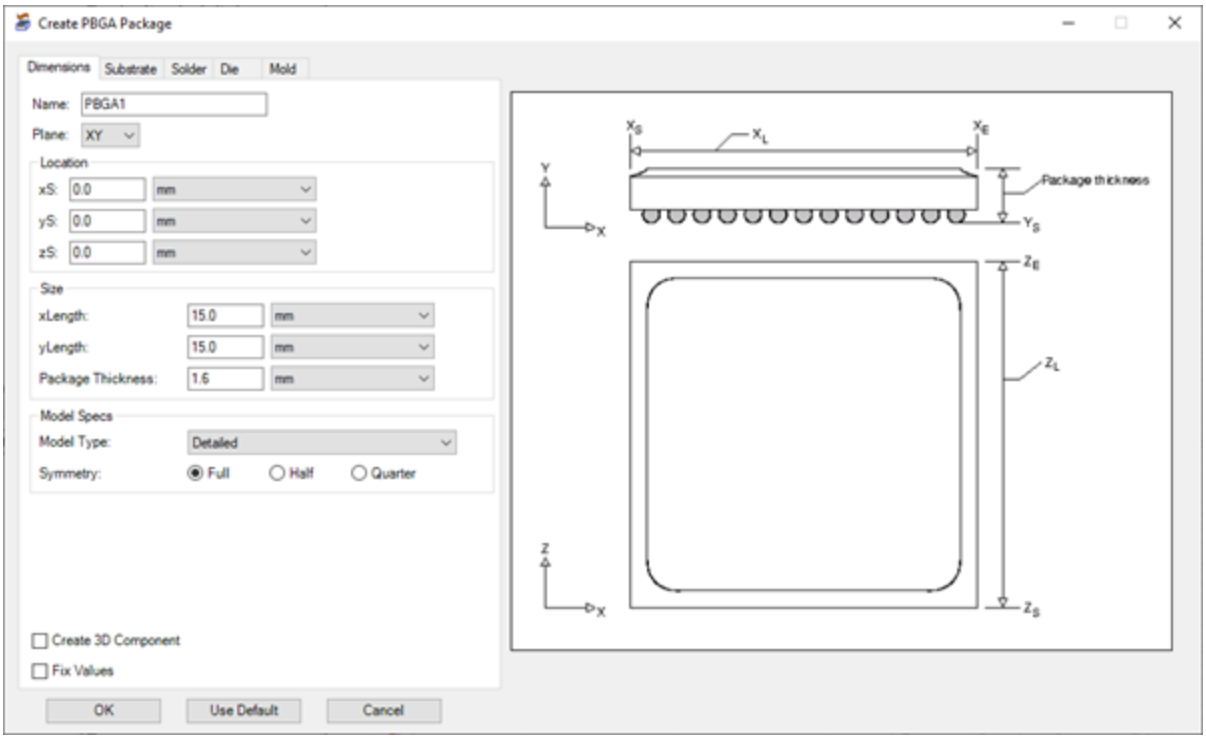
   Note: You can hide the grid by selecting **View** > **Grid Settings** and then selecting **Hide** in the **Grid Spacing** dialog box. Also, from the **View** > **Coordinate Systems** menu, you can hide the large coordinate triad and display a smaller coordinate triad in the bottom of the **3D Modeler** window.

2. From the **File** menu, select **Save As**, and save the project in the desired working directory.

Create a PBGA Package

The PBGA toolkit provides capabilities to create a standard Integrated-Circuit Plastic Ball Grid Array (PBGA) Package. You can find the 2D blueprints on the right side of most Geometry toolkits. Geometry Package toolkits provide the ability to specify dimensions and properties to create standard IC packages.

1. From the **Icepak** menu, select **Toolkit** > **Geometry** > **Packages** > **PBGA**.

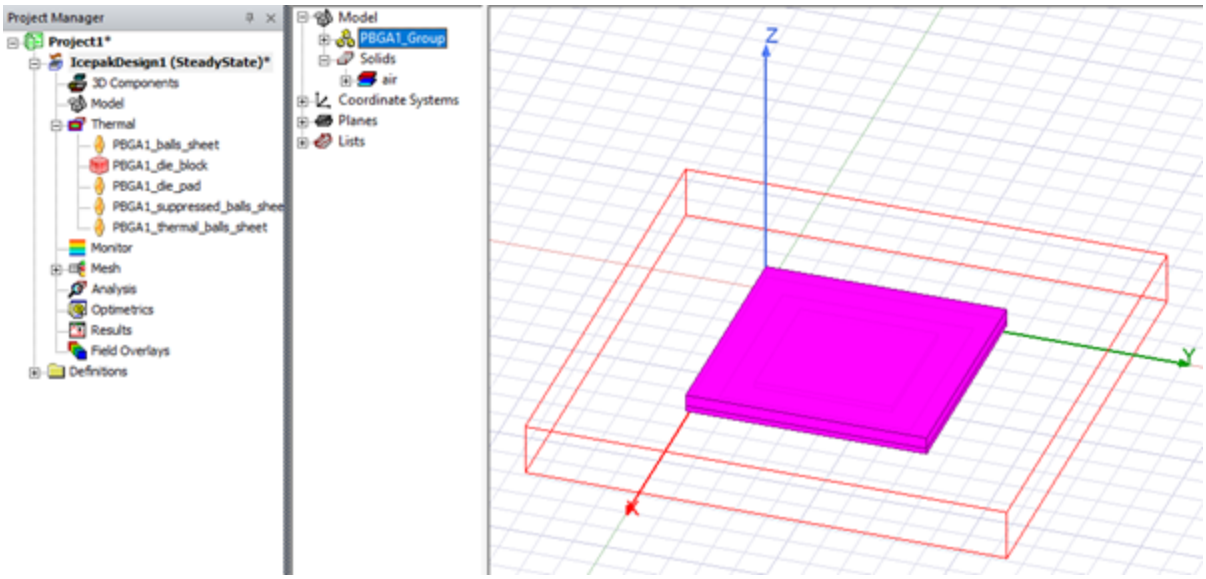


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Under **Model Specs**, change the **Model Type** to **Compact Conduction Model (CCM)**.

> **Note:**
>
> Most Icepak Geometry toolkits have a Compact Conduction Model (CCM) option that provides the ability to create a simplified version of the package. The solder ball array in PBGA is approximated as a solid covering the bottom surface of the package with equivalent uniform material properties. Additional approximations of this type are made leading to decreases in the complexity of the model. It is suggested that the user compare the detailed and CCM packages of a geometry toolkit.

3. On the **Solder** tab under **Distribution**, define the following settings:
   - Change the **Array Type** to **Peripheral Array**.
   - Change the **Number of Suppressed Rows** to **X** = 6 and **Y** = 6.
   - Change the **Number of Central Rows** to **X** = 4 and **Y** = 4.
4. On the **Die** tab under **Die Attach**, change the **Material** to **Si-Typical**.
5. Click **OK** to create the PBGA package.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

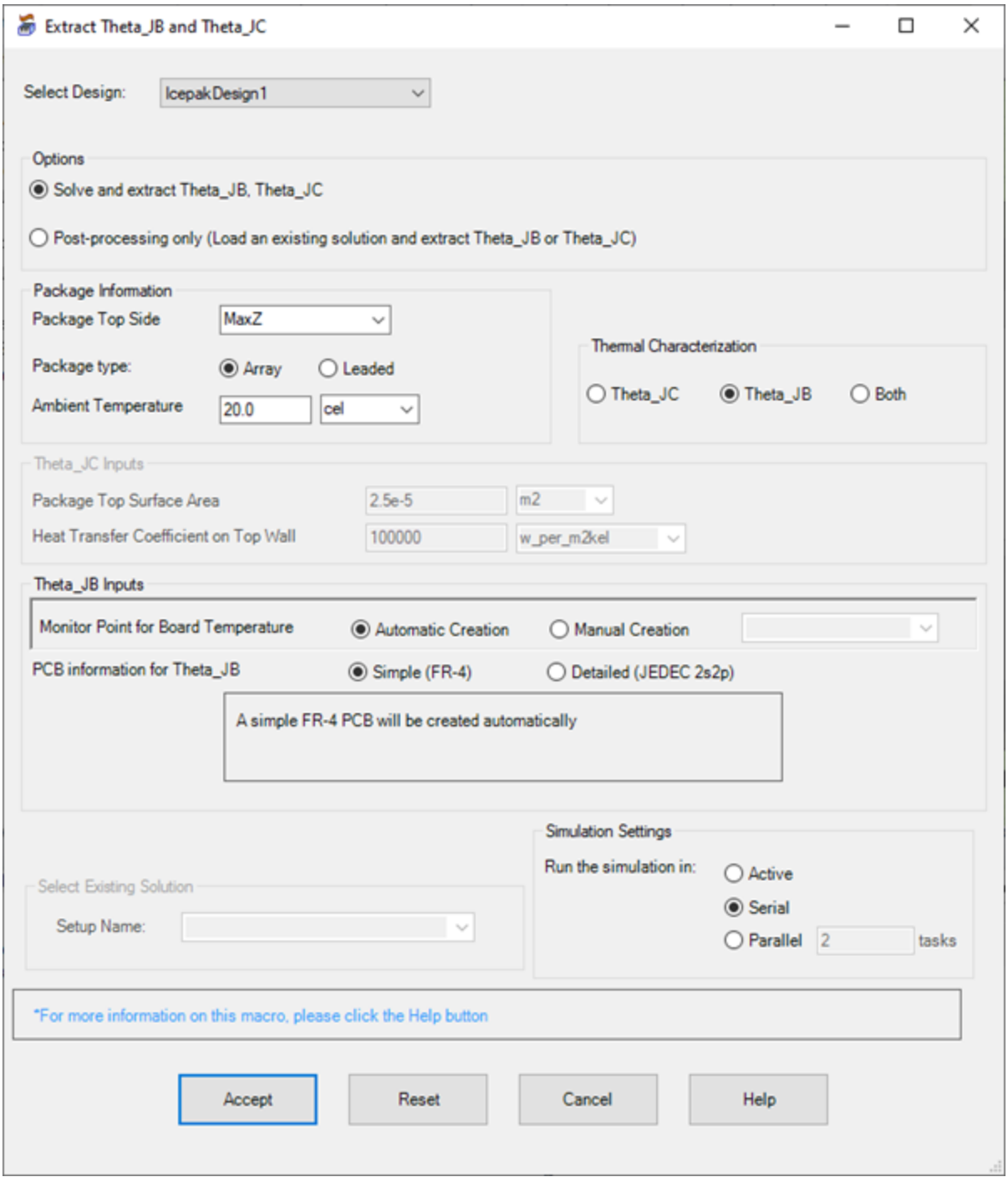
3 - Analyze and Extract Theta JB

Use the Extract Theta JB and Theta JC toolkit to solve the model determine the Theta Junction-to-board and/or Theta Junction-to-case value.

> **Note:**
>
> The toolkit also provides the ability to load an existing solution to determine the theta values for post-processing.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the **Icepak** menu, select **Toolkit** > **Modeling** > **IC Packages** > **Extract_JB_JC**.



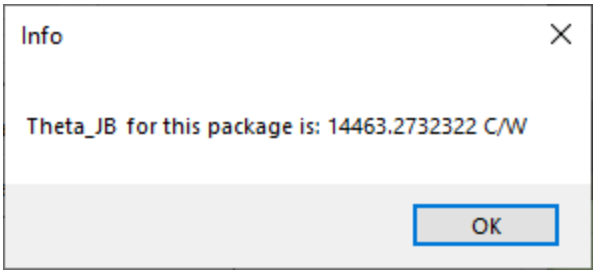
2. Under **Options**, ensure that **Solve and extract Theta_JB, Theta_JC** is selected.
3. Under **Package Information**, select **MaxZ** from the **Package Top Side** drop-down list.
4. For **Package Type**, ensure that **Array** is selected.
5. Under **Thermal Characterization**, select **Theta_JB**.

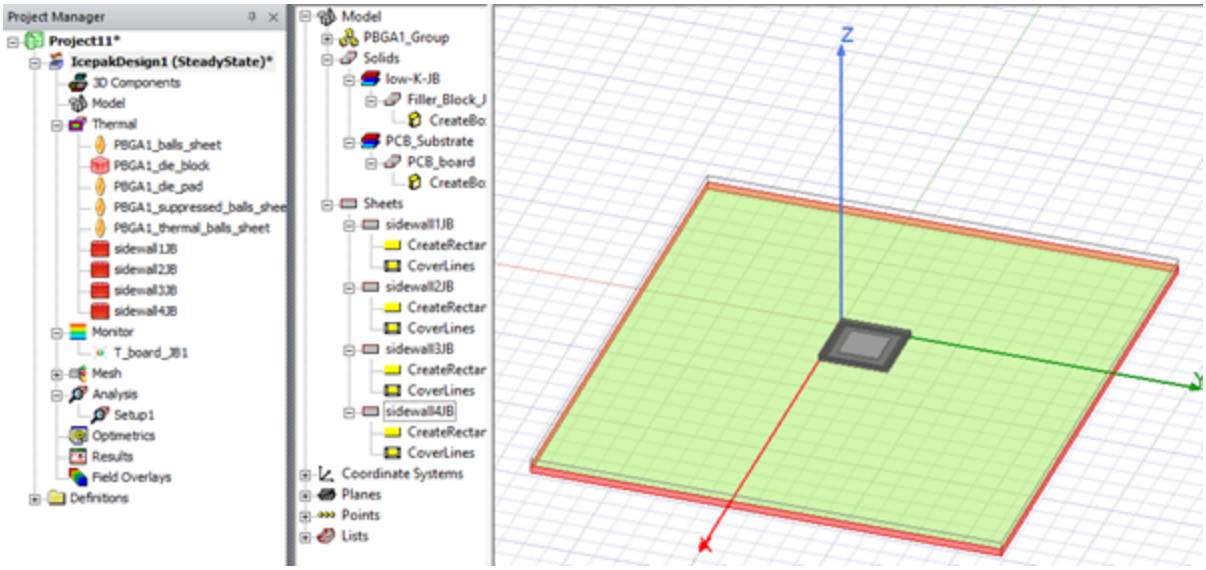
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Under **Theta_JB Inputs**, ensure that **Simple (FR-4)** is selected.

> **Note:**
>
> When using the Theta_JB radio option, the Extract Theta JB and Theta JC toolkit automatically creates a PCB under which the IC package sits. You can specify the type to be either a simple FR-4 material PCB or a detailed JEDEC 2s2p PCB.

7. Click **Accept** to start the analysis. When the analysis is complete, a dialog displays the theta JB value.





Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.