Chapter 19: Microelectronics Packages - Compact models

19.1. Introduction

This tutorial is a case study of a board design. A card supplier is making two package type changes to an existing commercial board. The objective of the thermal simulation project is to see if the selected new packages are likely to function without overheating. In the event of over heating, what kind of thermal management should be recommended?

In this tutorial, you will learn how to:

- Perform a board level simulation with appropriate package models.
- Determine if the selected new packages can function without overheating.

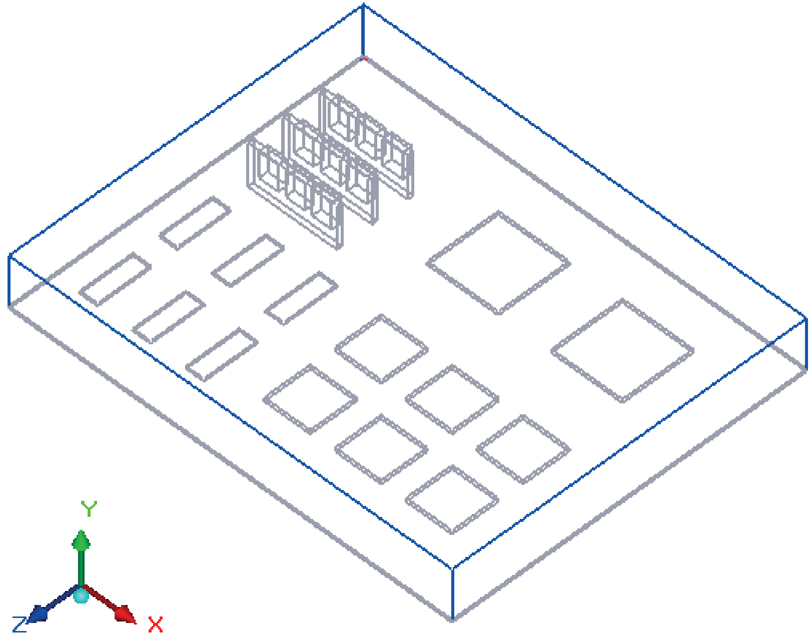
19.2. Prerequisites

This tutorial assumes that you have worked on Sample Session in the *Icepak User's Guide* and the first two ANSYS Icepak tutorials of this guide.

19.3. Problem Description

A designer is to select packages for a new design at the drawing board level. Available information about the board and packages is given. Determine cooling solutions in the event there is overheating.

Figure 19.1: Problem Specification



19.4. Step 1: Create a New Project

1. Copy the file `ICEPAK_ROOT/tutorials/compact-package/compact-package-modeling.tzr` to your working directory. You must replace *ICEPAK_ROOT* by the full path name of the directory where ANSYS Icepak is installed on your computer system.

2. Start ANSYS Icepak, as described in Starting ANSYS Icepak in the *Icepak User's Guide*.

3. Click **Unpack** in the **Welcome to Icepak** panel.

4. In the **File selection** panel, select the packed project file `compact-package-modeling.tzr` and click **Open**.

5. In the **Location for the unpacked project** file selection dialog, select a directory where you would like to place the packed project file, enter a project name (i.e., `test-1`) in the **New project** text field then click **Unpack**.

19.5. Step 2: Build the Model

This tutorial uses an existing model. ANSYS Icepak will display the model in the graphics window as shown in Figure 19.2: Layout of the board to be analyzed (p. 319). Available information about the board and packages is shown in Table 19.1: Available Details for Objects in the Model (p. 319) and Table 19.2: Available Information for 400 PBGA (p. 320).

Release 14.5 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.2: Layout of the board to be analyzed

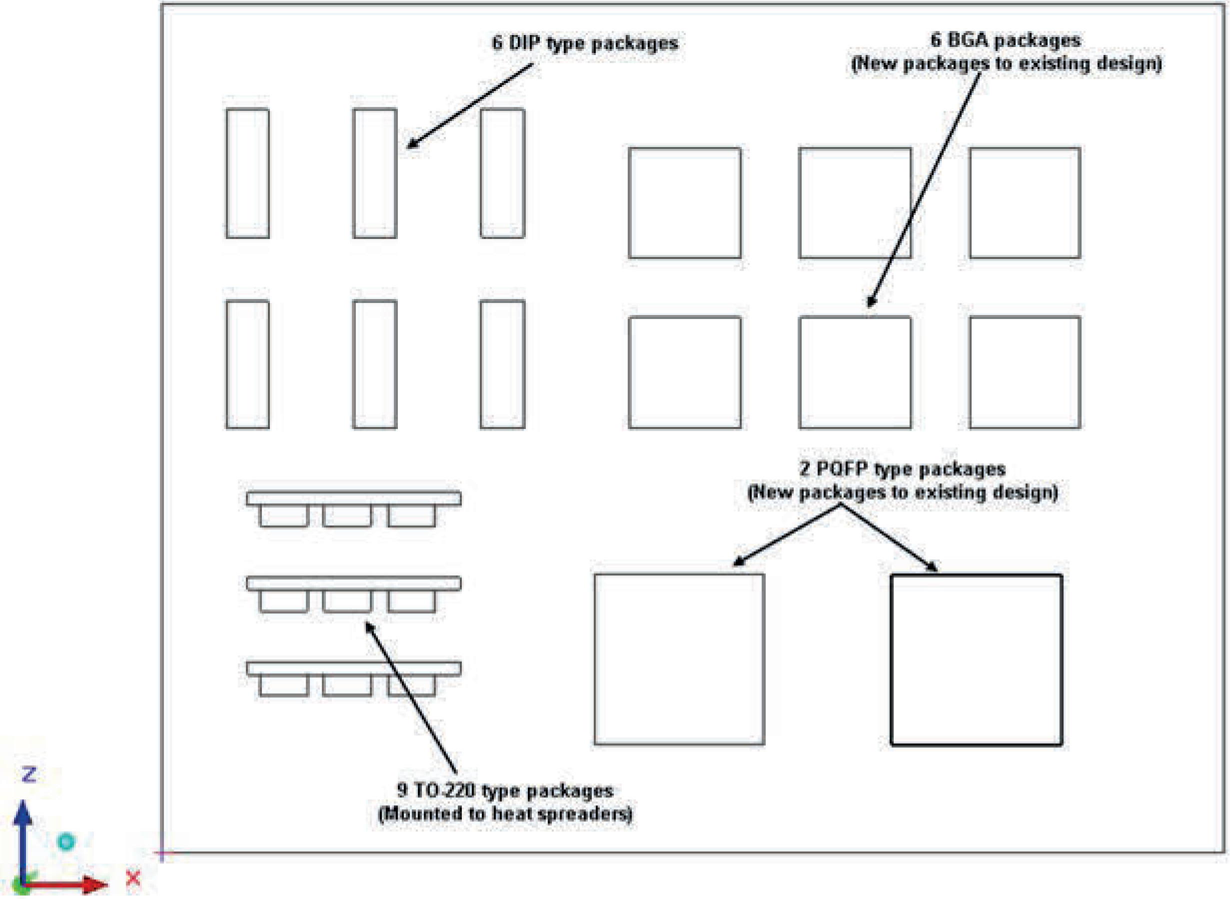


Table 19.1: Available Details for Objects in the Model

Object	# of Occurrences in model	Available information	Power (w)
PCB	1	1.6 mm thick, FR4 Material, six 1 oz. layers of Copper, 30% coverage for all layers	0
Heat Spreader for TO-220 packages	3	Extruded Aluminum	0
TO-220 Packages	9	θ_{jc} = 2.5° C/W	1.5
DIP	6	None	0.5
400 PBGA (new package type to the existing board)	6	See Table 19.2: Available Information for 400 PBGA (p. 320)	2.0

Release 14.5 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Object	# of Occurrences in model	Available information	Power (w)
232 PQFP (new package type to the existing board)	2	232 leads, 40 mm X 40 mm Footprint, 2 mm height	3.5

Note

An ounce of Copper is actually the thickness of 1 ounce/sq.ft of plane copper sheet. Using copper density this translates to a thickness of 0.035 mm.

Table 19.2: Available Information for 400 PBGA

Feature	Size (mm)	Material/Conductivity (W/mK)	Other info	Where to input this info?
Overall package	26 x 26 x 2.15			Dimensions tab
Mold compound		0.8		Die/Mold tab
Die	18 x 18 x 0.4	Silicon material		Die/Mold tab
Die Flag	18 x 18 x 0.035 (equivalent)	80.0 (effective)		Die/Mold tab
Die Attach	0.05 mm thick	Not mentioned		Die/Mold tab
Substrate	0.4 mm thick	FR4		Substrate tab
Substrate traces	0.035 mm thick	Copper	4 layers, top and bottom 30% coverage intermediate layers are 100% (plane layers)	Substrate tab
Vias	Unknown	Not mentioned	Number of vias unknown	Substrate tab (use 0 for vias)
Solder Balls	Standard	Solder	20 x 20 count, full array	Solder tab
Wire Bonds	Not mentioned	Usually Gold		Die/Mold tab

1. Create the PCB

   Create a PCB object by clicking on the **Create printed circuit boards** button ( ). Then edit the PCB by clicking the **Edit object** button ( ) while the PCB object is selected in the **Model** tree. Enter the following in the **Geometry** tab:

Object type	Name	Shape/Type/Plane	Global Coordinates (m) **XS— YS— ZS— XE— YE— ZE**
PCB	pcb.1	XZ	0.0 — 0.0 — 0.0— 0.25— NA— 0.2

   a. Go to the **Properties** tab. Enter the **PCB thickness** of 1.6 mm for **Substrate thickness**.

Release 14.5 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

以上内容仅为本文档的试下载部分，为可阅读页数的一半内容。如要下载或阅读全文，请访问：https://d.book118.com/348030035122006117