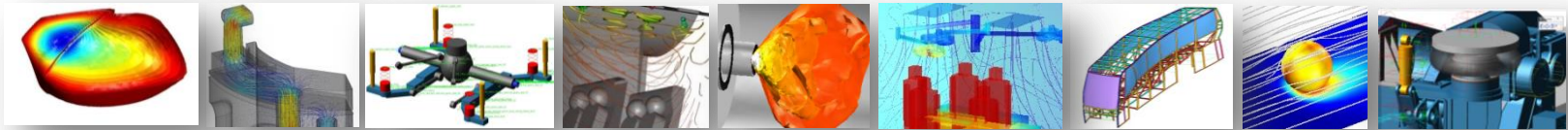

BE CAE & Test



A Comsol APP for thermal analysis of electronic devices

Giuseppe Petrone, Carmelo Barbagallo and Marco Scionti
BE CAE & TEST, Viale Africa 170 Sc. A, 95129 Catania (ITALY)

Grenoble, 14-16 October 2015





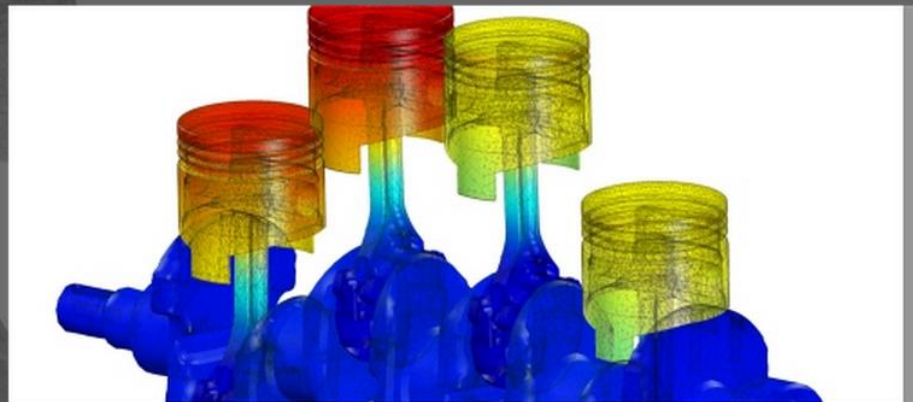
BE CAE & Test (<http://www.be-caetest.it>) provides consultancy services in several industrial sectors by using innovative **CAD/CAE modelling tools** and carrying out **experimental campaigns**

The company collaborates with **industrial partners** and **research centers** in several technologic fields

<http://www.be-caetest.it/>



- Activities
- Services
- Partners
- Customers
- News
- Contact us



CAE & Test

Our team of engineers and researchers supports companies and individuals during their products and processes development by using advanced CAD/CAE tools and organizing accurate experimental tests.

The experience gained in experimental vibro-acoustics, FEM and Multibody numerical simulation and the use of Multiphysics packages make BE CAE & Test the ideal partner to guarantee reliability, innovation and competitiveness of your products and processes.

COMSOL Certified Consultants

COMSOL Certified Consultants use their extensive experience with COMSOL Multiphysics and our related products to deliver turnkey solutions for a wide range of modeling projects. They produce ready-to-run models and reports with an in-depth analysis of the simulation results.

<https://www.comsol.it/certified-consultants>




India

- ATOA Scientific Technologies Pvt. Ltd.
- Noumenon

Israel

- Rotem Industries Ltd.

Italy

- BE CAE & Test

Luxembourg (BeNeLux)

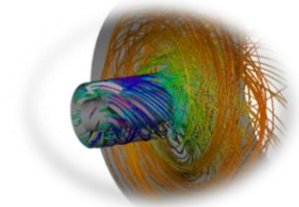
- Siegel Schleimer Ingénieurs-conseils s.à r.l.

Our team of engineers and researchers supports companies and individuals during their products and processes development by using advanced CAD/CAE tools and organizing accurate experimental tests.

The experience gained in experimental vibro-acoustics, FEM and Multibody numerical simulation and the use of Multiphysics packages make BE CAE & Test the ideal partner to guarantee reliability, innovation and competitiveness of your products and processes.

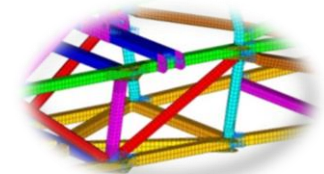
➤ **Fluid dynamics and thermal analyses**

- Environmental energetics (HVAC, thermal comfort, IAQ)
- Industrial energetics (Thermal design, energy conversion, reacting flows)



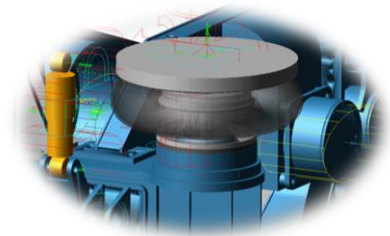
➤ **Structural analyses**

- Linear and non-linear statics, dynamic and vibro-acustics analyses in industrial and civil applications



➤ **System dynamics and Multi-Body analyses**

- Vehicle and rail dynamics (handling, ride comfort)
- Kinematics, dynamics, rigid and flexible bodies analyses of mechanisms

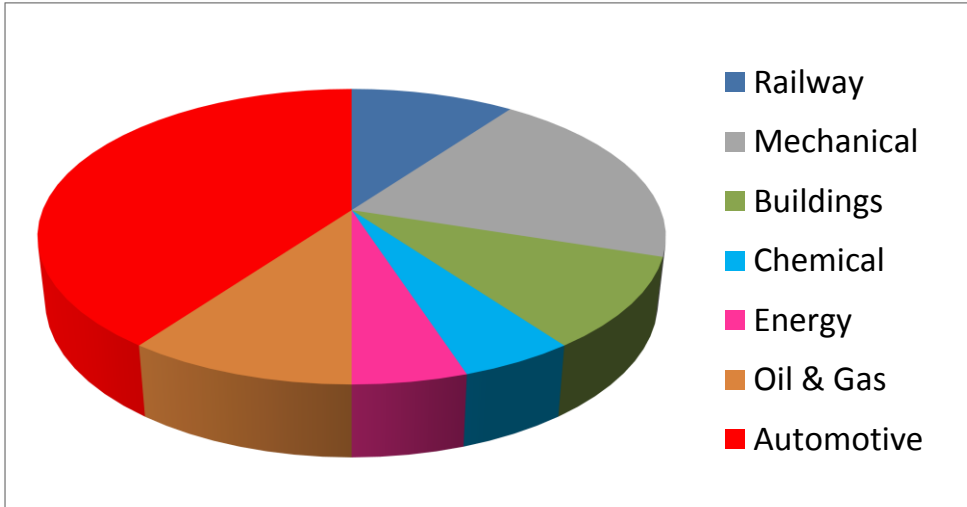


➤ **Experimental testing**

- Ride comfort (NVH), modal analyses
- Human body vibrations (ISO standard)



BE CAE & Test / Business areas and Customers



BE CAE & Test collaborates with world-wide companies active in the fields of numerical simulations and experimental measurements.



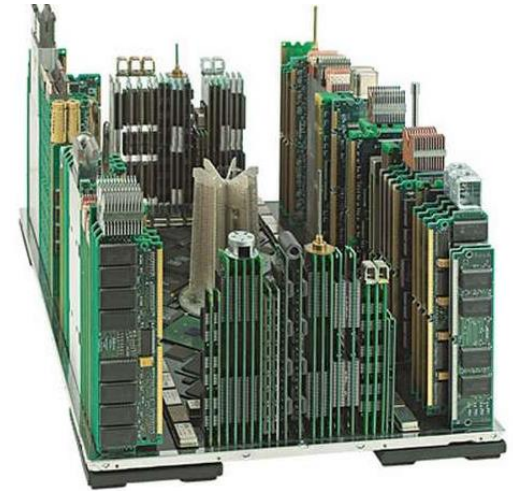
➤ **Why thermal analysis of electronic devices?**

- Electronic devices produce a **very high rate** of specific **heat** (small dimensions)
- Exceeding in maximum safe operating temperature means strong reduction of **efficiency, reliability** and **lifetime**

Overheating is one of the main causes of failure for electronic equipment



Thermal design represents an unavoidable step in pre-production phase **in order to ensure reliability and performance of the electronic devices**



<http://www.fludit.com/images/stories/Design/Cities-Made-From-Electronic-Components>

➤ Why thermal analysis of electronic devices?

- Electronic devices produce a **very high rate of specific heat** (small dimensions)
- Exceeding in maximum safe operating temperature means strong reduction of **efficiency, reliability and lifetime**

Overheating is one of the main causes of failure for electronic equipment

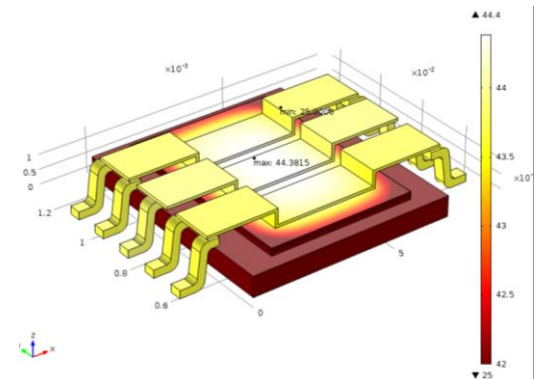
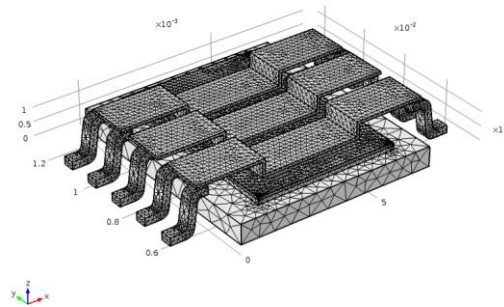
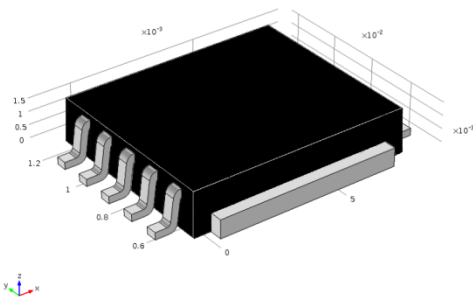


Thermal design represents an unavoidable step in pre-production phase in order to ensure reliability and performance of the electronic devices



<http://www.fludit.com/images/stories/Design/Cities-Made-From-Electronic-Components>

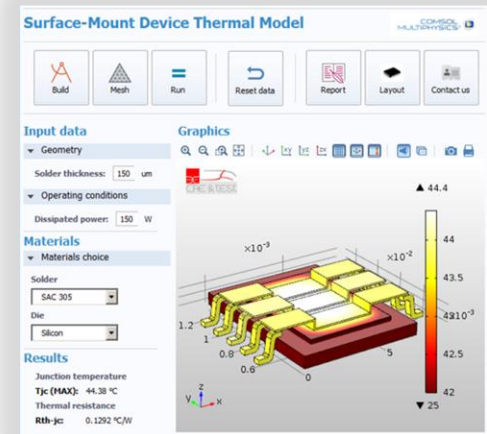
MODELLING & SIMULATION



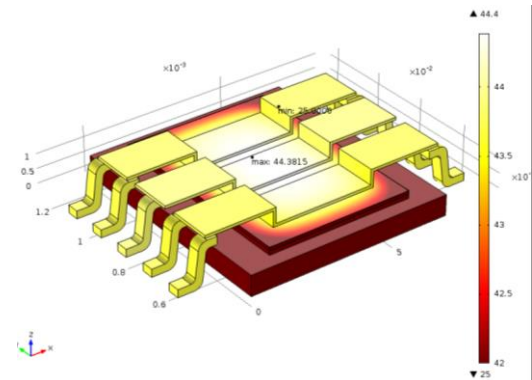
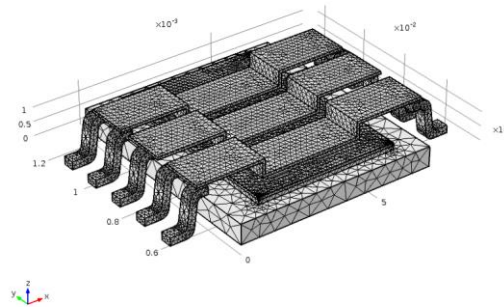
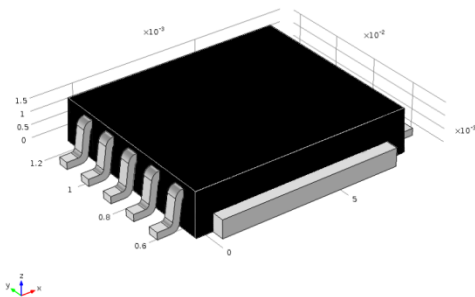
- Among the **new features** introduced in **Comsol Multiphysics**, the opportunity of building up **customized GUI** by exploiting the **Application Builder** is for sure one of the most promising

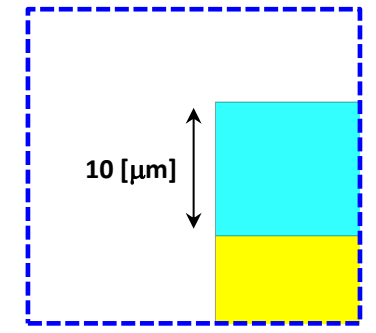
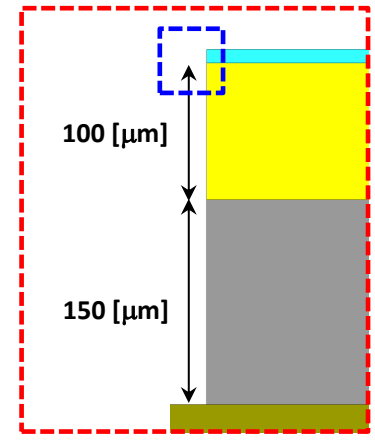
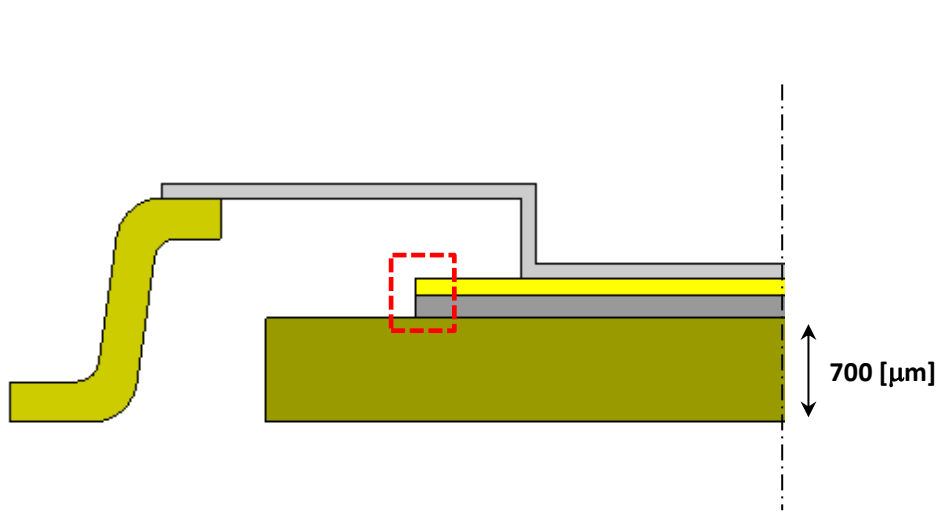
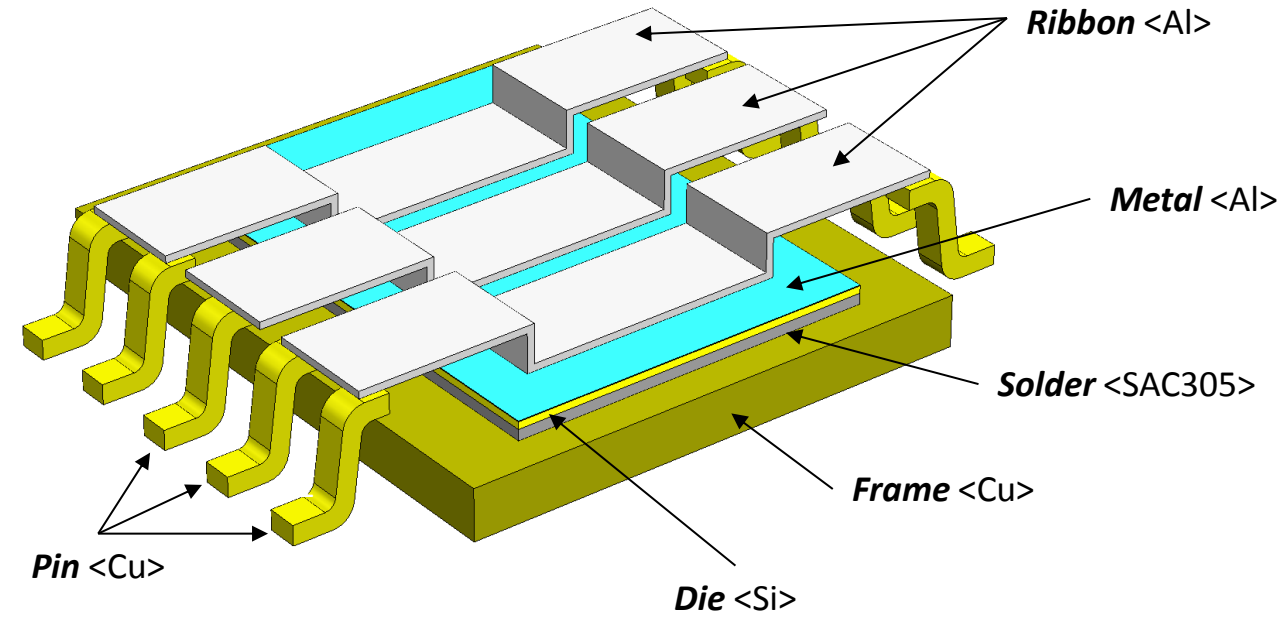


Arising from this concept, BE CAE & Test built-up a Comsol APP to analyse the thermal behavior of an electronic Surface-Mount Device (SMD)



FROM MODEL TO APP





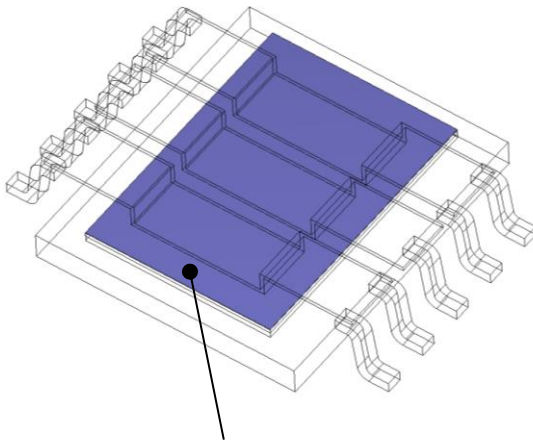
THERMAL ANALYSIS

Input: Dissipated thermal power
 Frame temperature (corresponding to initial conditions)

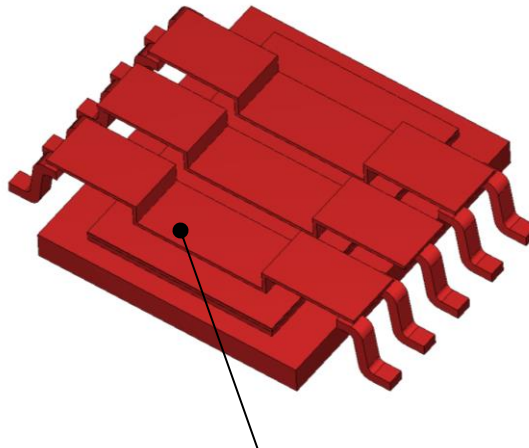
Output: Temperature distribution

Energy equation

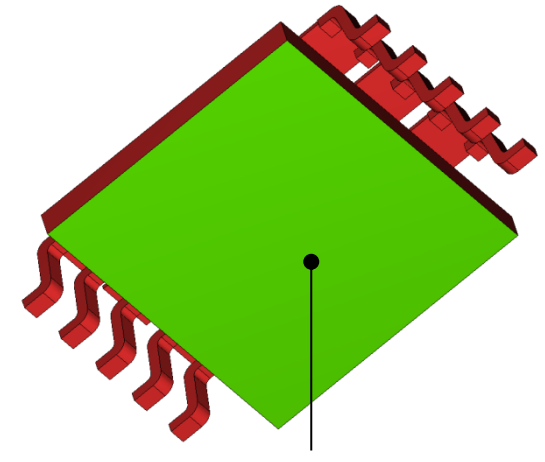
$$\rho C_p \frac{\partial T}{\partial t} = \nabla \cdot (k \nabla T) + Q$$



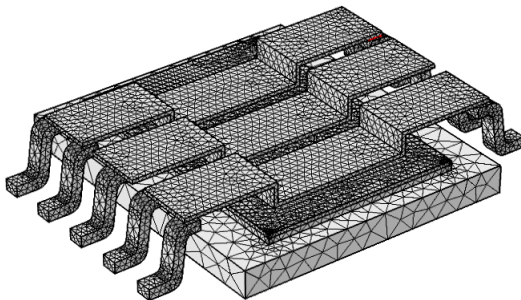
Heat source (Die ON)
 $P_{th} = 150 \text{ W}$



Insulation (package)
 $-n \cdot k \nabla T = 0$



Frame temperature
 $T_{ref} = 25 \text{ }^\circ\text{C}$



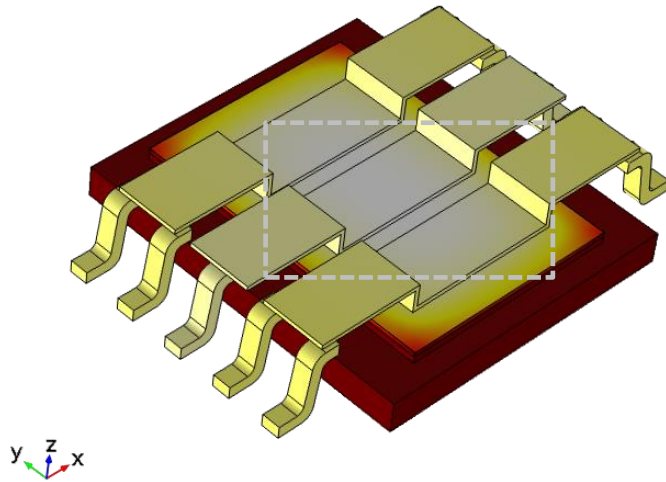
- Non-uniform and non-structured computational grids made of tetrahedral Lagrange elements of order 2 (~ 45,000 elements)
- Time-marching by using a Implicit Differential-Algebraic (IDA) solver based on a variable-order and variable-step-size Backward Differentiation Formulas (BDF)
- Direct solver (PARDISO) for linear systems
- Computational node : 2 x 64-bit dual-core @2,30 GHz - RAM 128 GB

THERMAL ANALYSIS

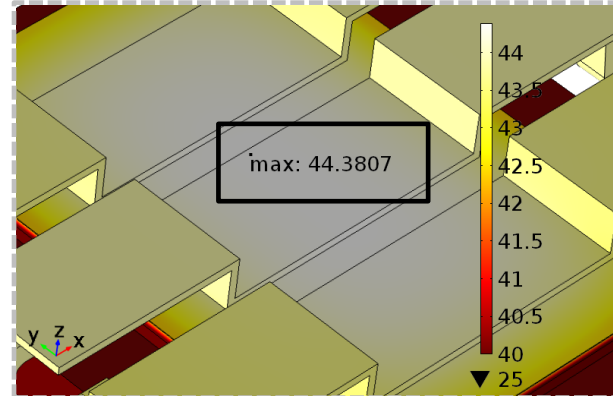
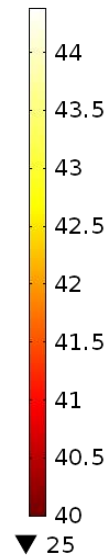
Input: Dissipated thermal power
 Frame temperature (corresponding to initial conditions)

Output: Temperature distribution

Temperature [°C]



▲ 44.4

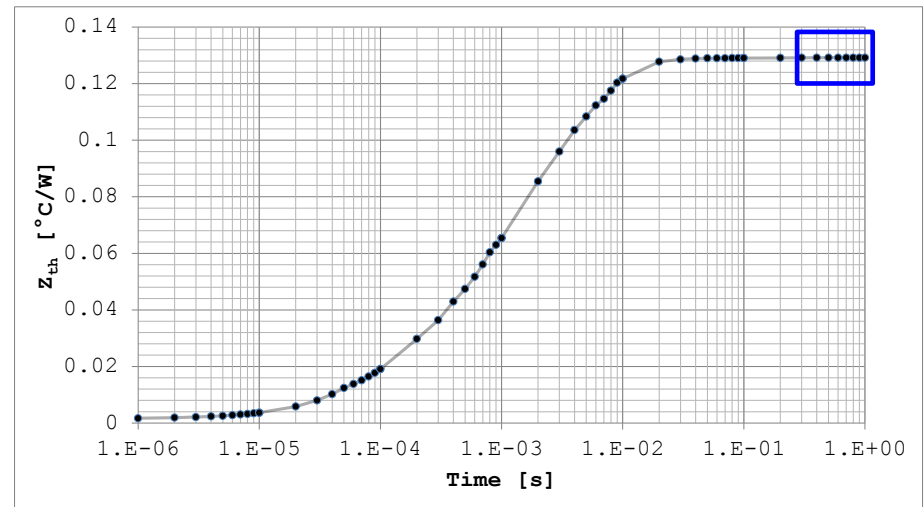


$T_{jc} (MAX) = 44,4 [^{\circ}C]$

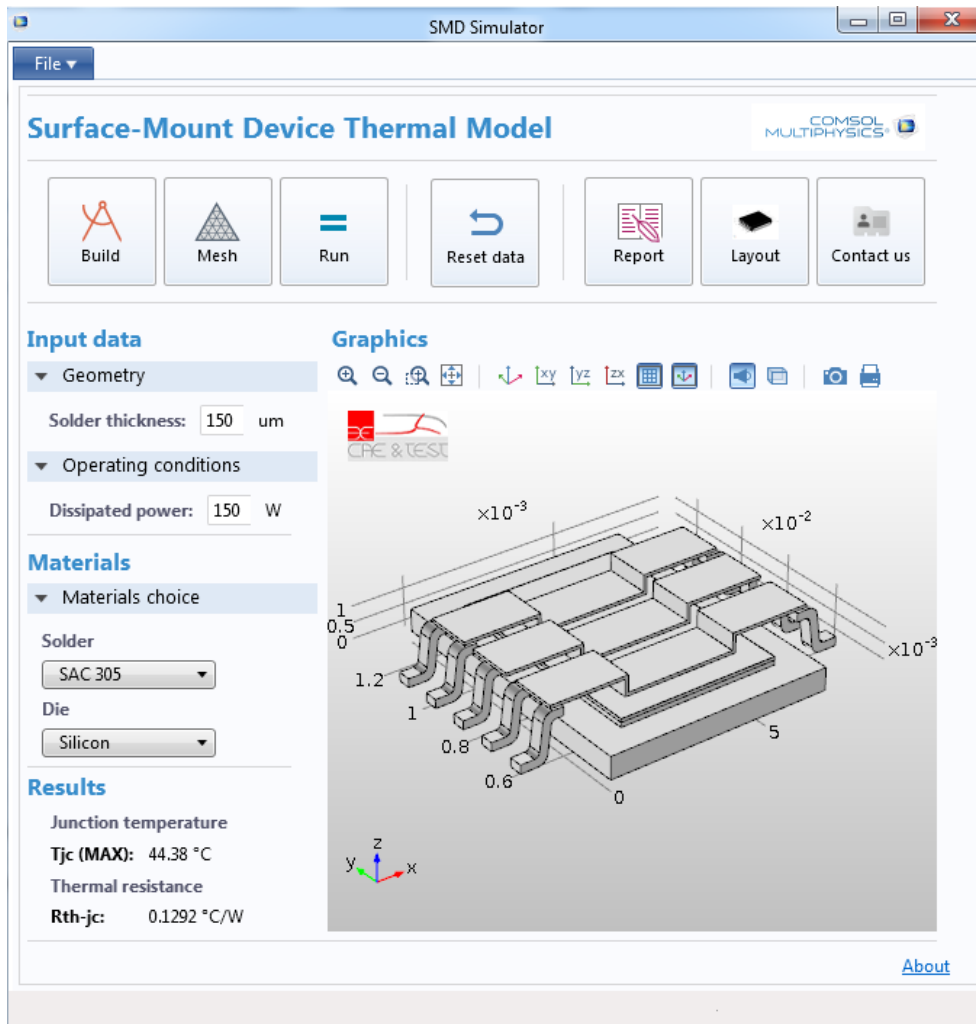
$R_{th-jc} = 0,13 [^{\circ}C/W]$

Thermal Impedance

$$Z_{th} = \frac{T - T_{ref}}{P_{th}}$$



Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”



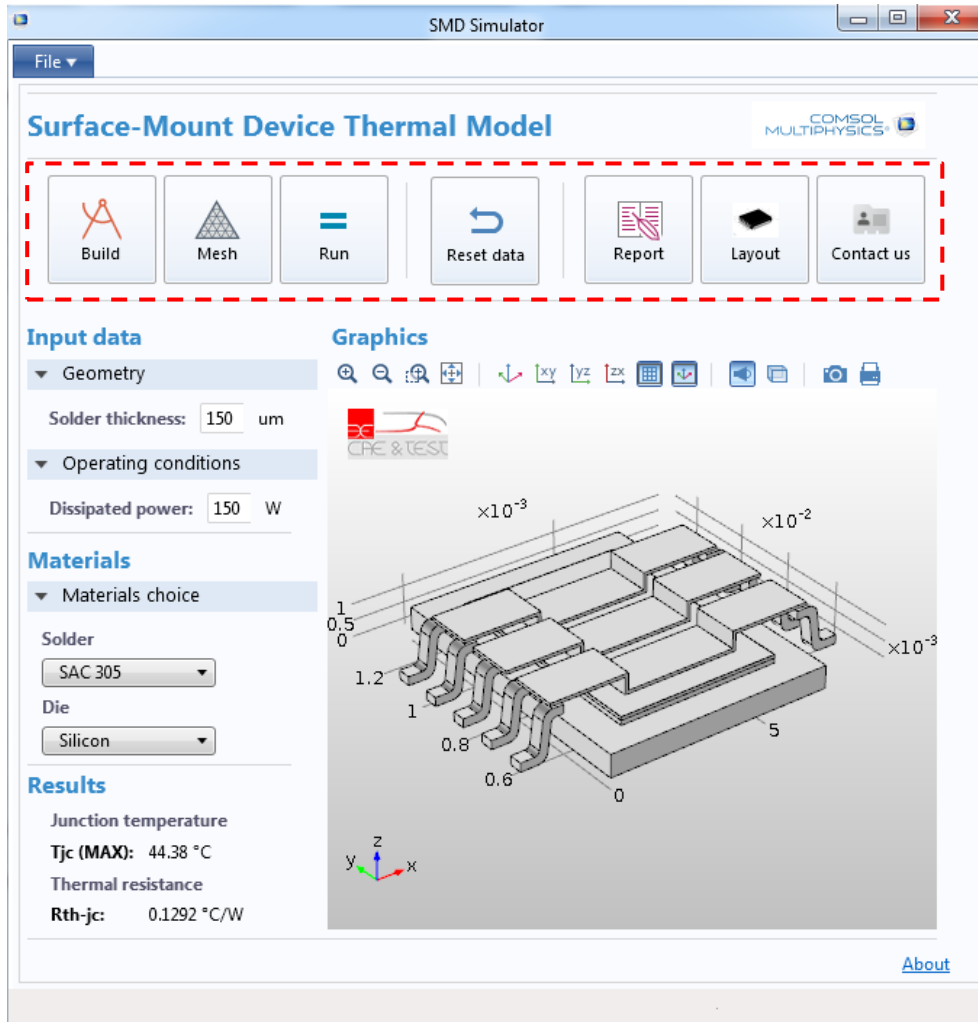
➤ What is that?

- Customized GUI allowing users to carry-out parametrical simulations without build models

➤ Which kind of «paramteric» analysis?

- **Geometrical**
- **Constitutive:** materials, assumption (i.e. plasticity model in structural analysis, flow regime in fluid dynamics, ...)
- **Funcional:** any operational or working condition
- **Derived value:** any value derived from FE dependent variable solved (i.e. a thermal flux from temperature solution in thermal analysis)
- ...
- ...

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”



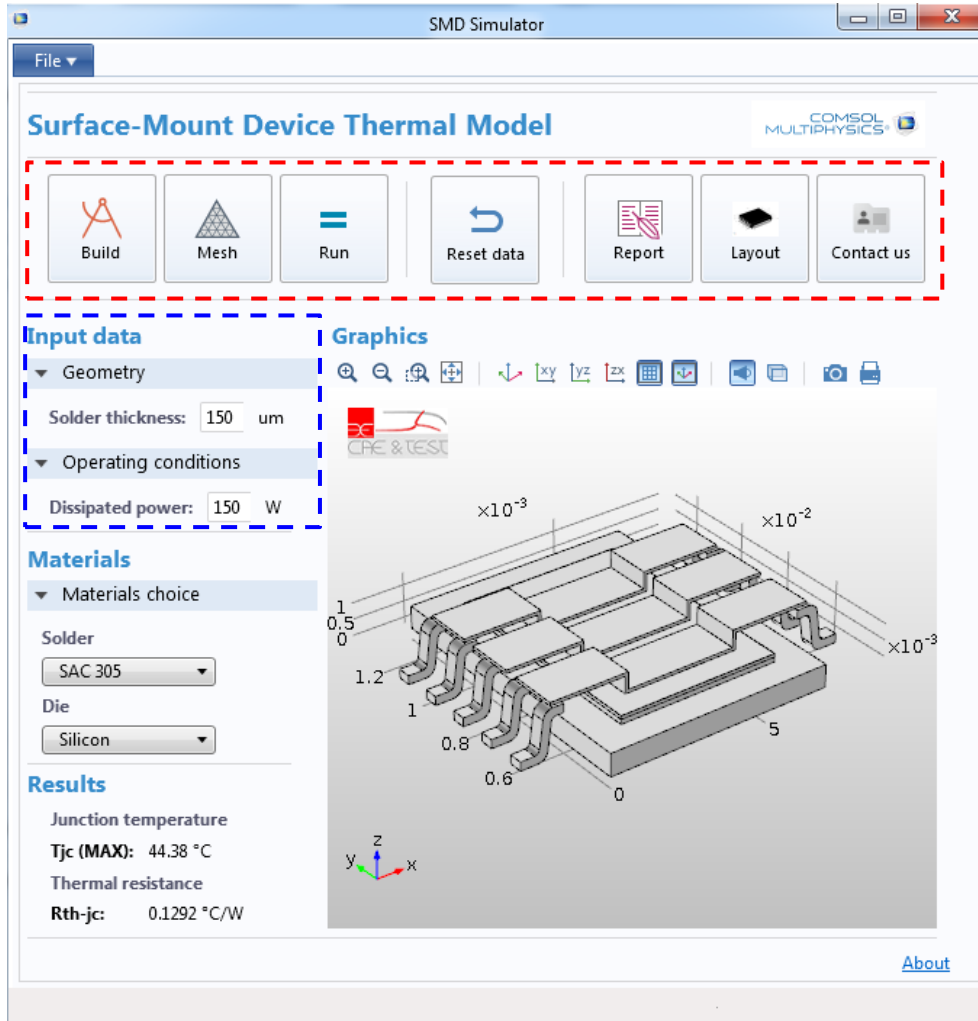
The screenshot displays the COMSOL SMD Simulator interface. The title bar reads "SMD Simulator". The main window is titled "Surface-Mount Device Thermal Model" and includes the COMSOL MULTIPHYSICS logo. A red dashed box highlights a row of action buttons: Build, Mesh, Run, Reset data, Report, Layout, and Contact us. Below this, the interface is divided into several sections:

- Input data:**
 - Geometry: Solder thickness: 150 μm
 - Operating conditions: Dissipated power: 150 W
- Materials:**
 - Materials choice:
 - Solder: SAC 305
 - Die: Silicon
- Results:**
 - Junction temperature
 - $T_{jc} \text{ (MAX): } 44.38 \text{ } ^\circ\text{C}$
 - Thermal resistance
 - $R_{th-jc}: 0.1292 \text{ } ^\circ\text{C/W}$

The central Graphics area shows a 3D model of a surface-mount device with dimensions. The x-axis ranges from 0 to 5 (scaled by $\times 10^{-3}$), the y-axis from 0 to 1.2 (scaled by $\times 10^{-3}$), and the z-axis from 0 to 1 (scaled by $\times 10^{-2}$). A coordinate system (x, y, z) is shown at the bottom left of the graphics area. An "About" link is visible at the bottom right of the interface.

➤ Action button

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”



The screenshot displays the COMSOL SMD Simulator interface for a "Surface-Mount Device Thermal Model". The window title is "SMD Simulator".

Top Panel (Action buttons): A row of seven buttons: Build, Mesh, Run, Reset data, Report, Layout, and Contact us. These buttons are enclosed in a red dashed box.

Input data (Input field): A section on the left with a blue dashed border containing:

- Geometry: Solder thickness: um
- Operating conditions: Dissipated power: W

Materials: A section on the left containing:

- Materials choice: Solder (SAC 305), Die (Silicon)

Results: A section on the left showing:

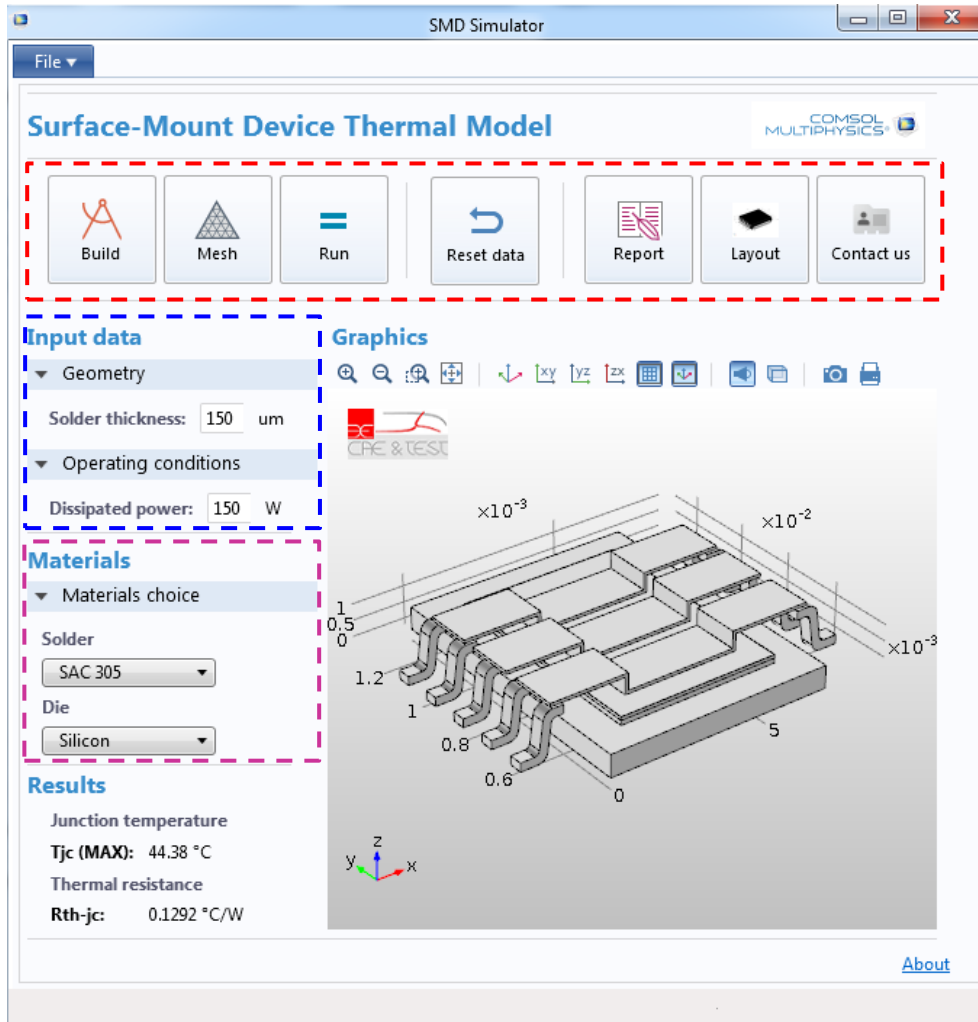
- Junction temperature: $T_{jc} (MAX): 44.38 \text{ } ^\circ\text{C}$
- Thermal resistance: $R_{th-jc}: 0.1292 \text{ } ^\circ\text{C/W}$

Graphics: A 3D model of the SMD assembly is shown in the center. The model includes a die, solder, and a substrate. Dimensions are indicated: $\times 10^{-3}$ for the die width and $\times 10^{-2}$ for the solder length. A coordinate system (x, y, z) is visible at the bottom left of the graphics area.

➤ Action button

➤ Input field

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”



Surface-Mount Device Thermal Model

COMSOL MULTIPHYSICS

Action buttons: Build, Mesh, Run, Reset data, Report, Layout, Contact us

Input data:

- Geometry
 - Solder thickness: um
- Operating conditions
 - Dissipated power: W

Materials:

- Materials choice
 - Solder:
 - Die:

Results:

- Junction temperature
- T_{jc} (MAX): 44.38 °C
- Thermal resistance
- R_{th-jc}: 0.1292 °C/W

3D Model Dimensions:

- Length: 1.2 × 10⁻³
- Width: 0.8 × 10⁻³
- Height: 0.6 × 10⁻³
- Distance between pads: 1 × 10⁻³
- Distance from edge: 0.5 × 10⁻³
- Distance from edge: 1 × 10⁻²
- Distance from edge: 5 × 10⁻³

Coordinate System: x, y, z

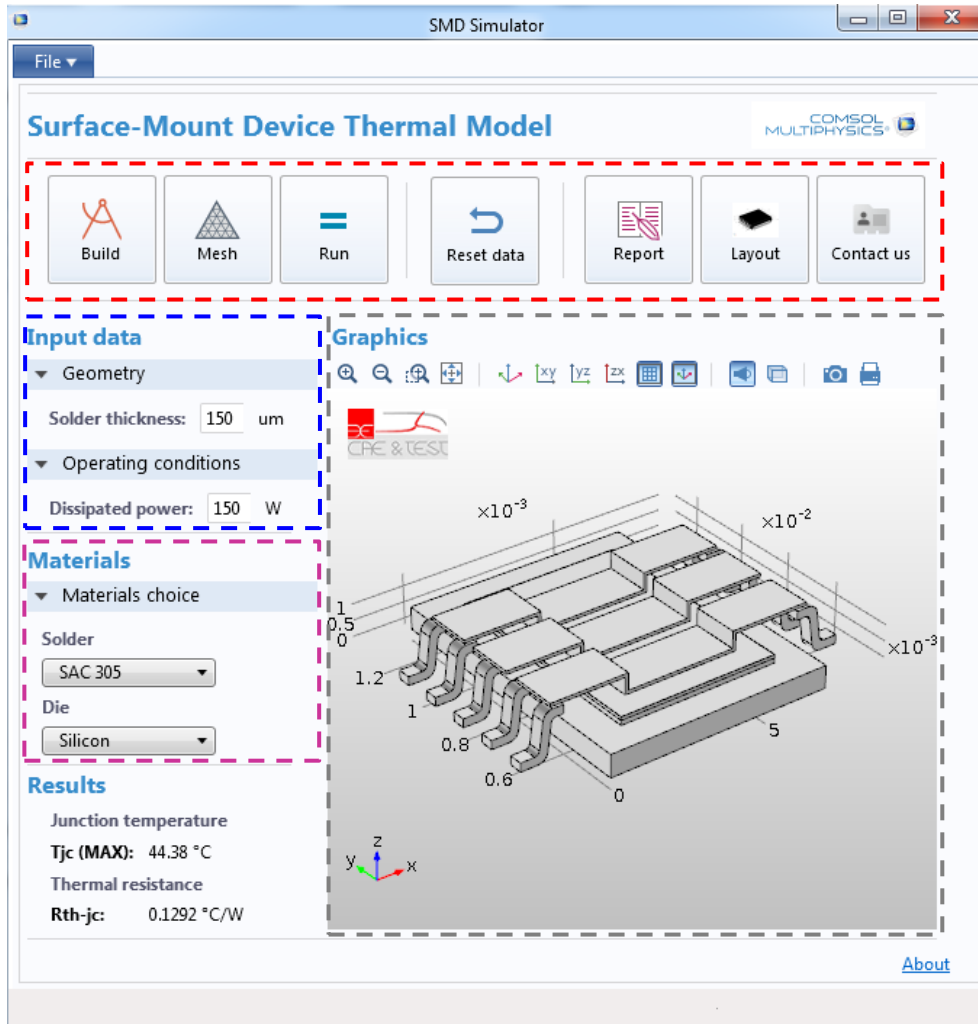
About

➤ Action button

➤ Input field

➤ Combo Box

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”



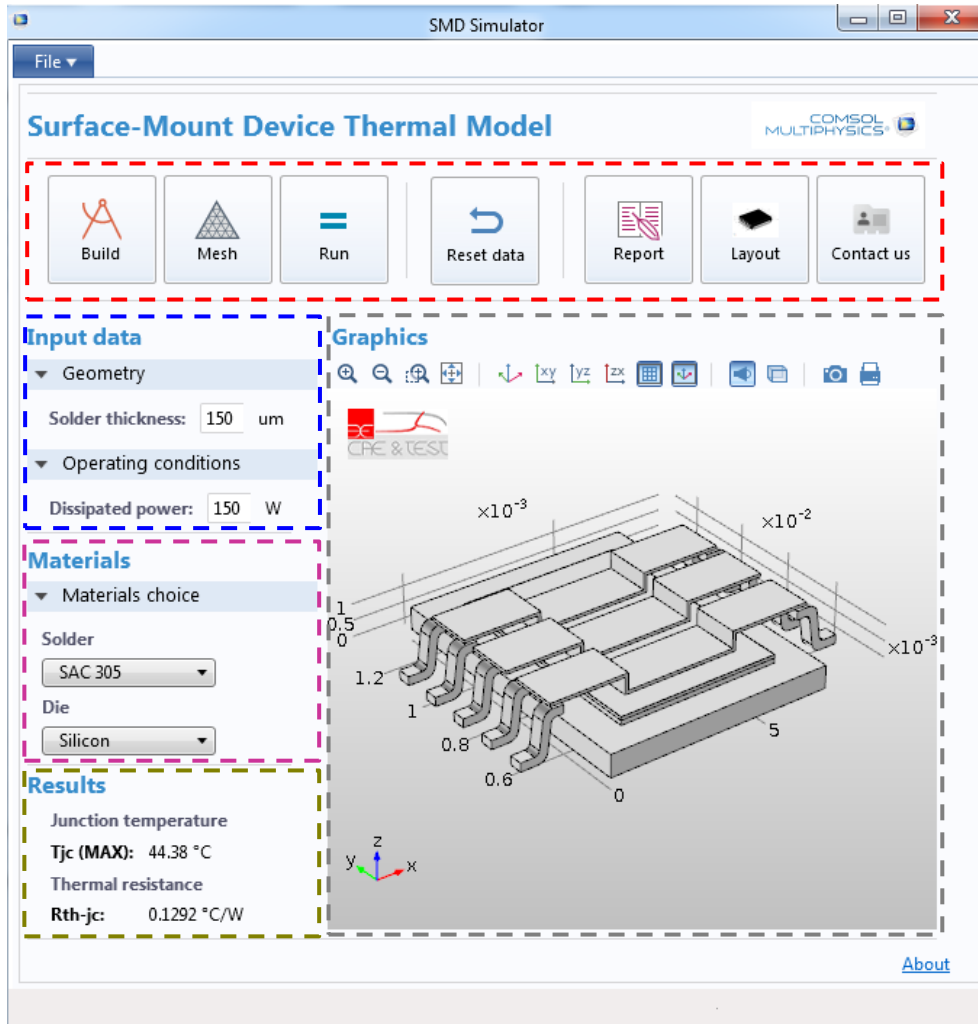
➤ Action button

➤ Input field

➤ Combo Box

➤ Graphics

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”



➤ Action button

➤ Input field

➤ Combo Box

➤ Graphics

➤ Data Display

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”

➤ Geometry - > Input field

The screenshot displays the COMSOL Multiphysics interface for a thermal analysis of an SMD. The **Model Builder** on the left shows the project structure, with a red arrow pointing to the **Parameters** node under **Global Definitions**. The **Parameters** table in the center lists the following data:

Name	Expression	Value	Description
s_solder	150[um]	1.5E-4 m	Solder thickness
s_frame	7.0E-4[m]	7E-4 m	Frame thickness
P_max	150[W]	150 W	Dissipated power
T_ref	25[degC]	298.15 K	Reference temperature

A red dashed box highlights the **Parameters** table, with the text *Declaration of parameters* written below it. The **Graphics** window on the right shows a 3D model of the SMD assembly with dimensions: 1.2, 0.8, 0.6, 1, 0.5, 0, 1, 5, and 0. Scale factors of $\times 10^{-3}$ and $\times 10^{-2}$ are indicated. The **Messages** window at the bottom right shows the text: "COMSOL 5.1.0.136 Opened file: Mod_App_09.mph".

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”

➤ Geometry - > Input field

The screenshot displays the COMSOL Multiphysics software interface. The main window is titled "Mod_App_09.mph - COMSOL Multiphysics". The top menu bar includes File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, and Results. The ribbon contains various toolbars for Application Builder, Model, Definitions, Geometry, Materials, Physics, Mesh, Study, Results, and Layout.

The **Model Builder** tree on the left shows the project structure:

- Mod_App_09.mph (root)
 - Global Definitions
 - Parameters
 - Variables
 - Materials
 - SMD (Surface-Mount Device) (comp1)
 - Definitions
 - Geometry
 - importa 1 (imp1)
 - Estrusione - solder (ext1)** (highlighted with a red arrow)
 - Baseplate wp (wp1)
 - Estrusione - baseplate (ext2)
 - Forma l'unione (fin)
 - Materials
 - Heat transfer in solids (ht)
 - Mesh 1
 - Analysis
 - Resultati

The **Properties** window for the **Extrude** feature is open, showing the following settings:

- Label: Estrusione - solder
- General: Extrude from: Faces
- Input faces: imp1 (92)
- Active: OFF
- Unite with input objects: OFF
- Distances from Plane:
 - Distances (m): s_solder
- Parametric design: *Parametric design*
- Keep cross-sectional faces: ON
- Reverse direction: OFF

The **Graphics** window on the right shows a 3D model of the SMD component. The model is a rectangular baseplate with several raised rectangular features. The dimensions are indicated as $\times 10^{-3}$ and $\times 10^{-2}$. The axes are labeled x, y, and z. The model is shown in a perspective view.

The **Messages** window at the bottom right shows the following text:


```
COMSOL 5.1.0.136
Opened file: Mod_App_09.mph
```

The system tray at the bottom of the screen shows the date and time as 11:42, and the system resources as 1.17 GB | 1.33 GB.

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”

➤ Constitutive / Materials -> Combo Box

The screenshot displays the COMSOL Multiphysics interface for a thermal analysis of an SMD. The **Model Builder** on the left shows a tree structure where the **Materials** folder under the **SMD (Surface-Mount Device)** component is highlighted with a red dashed box. A red arrow points to the **Link - solder (matLink1)** material entry. The **Properties** panel on the right shows the **Material Link** settings for this material, including the **Geometric Entity Selection** (Domain, Solder) and the **Material Contents** table.

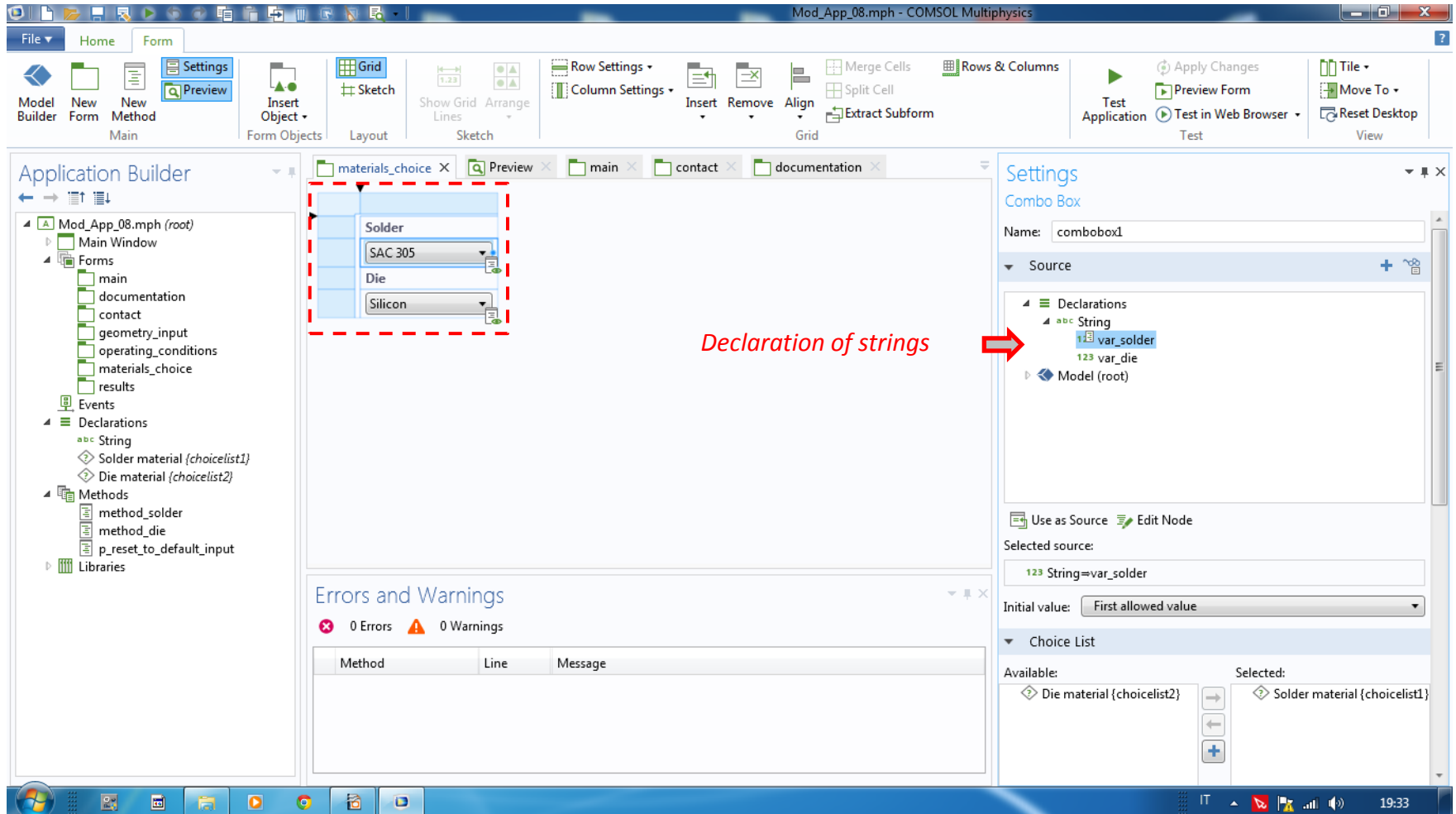
Property	Name	Value	Unit	Property
✓ Density	rho	7370	kg/m ³	Proprietà
✓ Thermal conductivity	k	75	W/(m·K)	Proprietà
✓ Heat capacity at constant pres...	Cp	220	J/(kg·K)	Proprietà

The **Graphics** window on the right shows a 3D model of the SMD assembly with dimensions in meters (e.g., 1.2, 0.8, 0.6, 1, 0.5, 1, 5, 10⁻³, 10⁻², 10⁻³). The **Messages** window at the bottom indicates the file **Mod_App_08.mph** was opened.

Global material / local link

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”

➤ Constitutive / Materials -> Combo Box



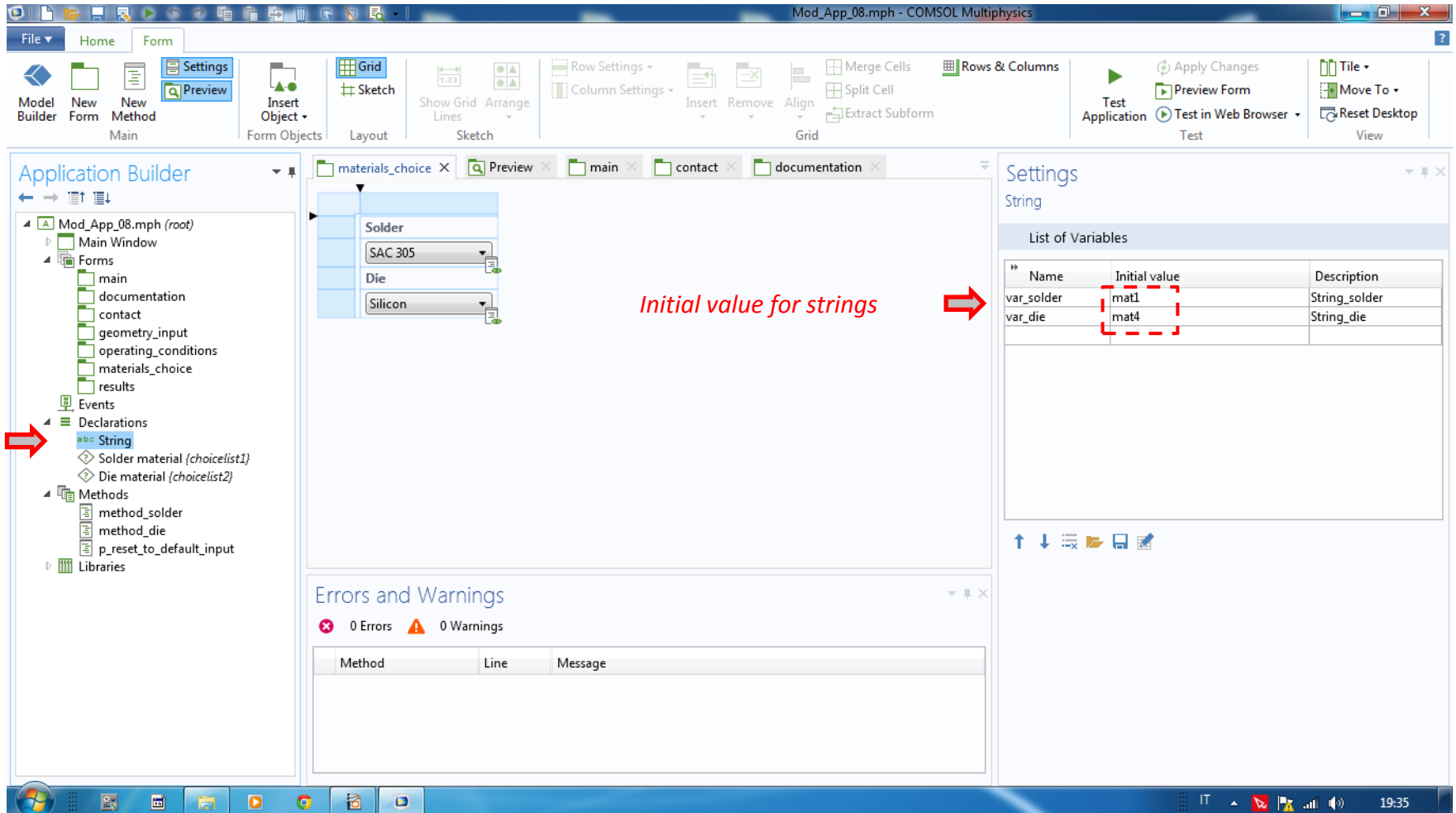
The screenshot displays the COMSOL Multiphysics interface for building an application. The main window shows a tree view of the application structure, including a 'materials_choice' component. The 'Settings' window for the 'Combo Box' component is open, showing the following configuration:

- Name:** combobox1
- Source:**
 - Declarations:
 - String:
 - 123 var_solder
 - 123 var_die
- Selected source:** 123 String=var_solder
- Initial value:** First allowed value
- Choice List:**
 - Available: Die material {choicelist2}
 - Selected: Solder material {choicelist1}

Declaration of strings

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”

➤ Constitutive / Materials -> Combo Box



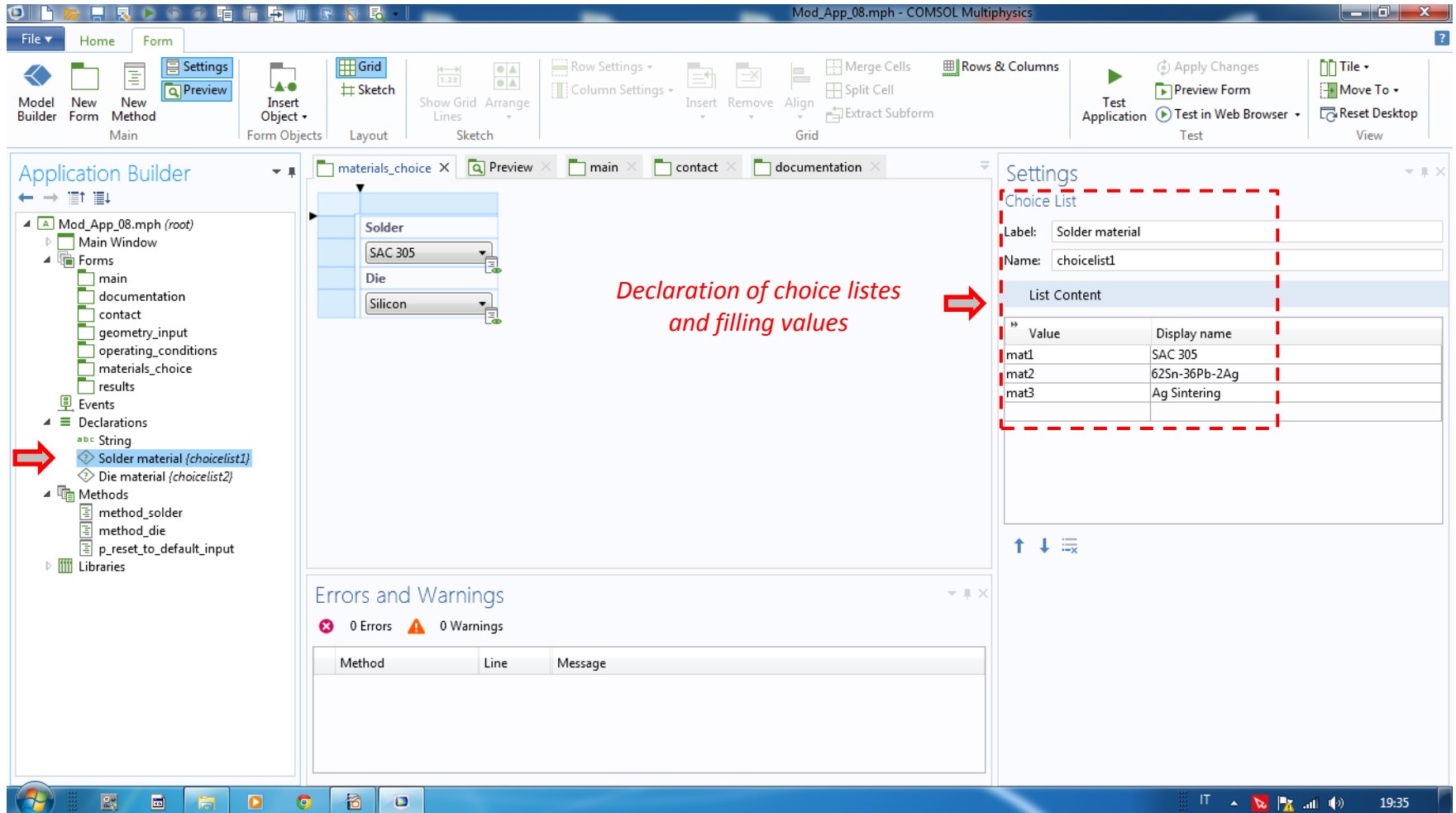
The screenshot shows the COMSOL Multiphysics interface for building an application. The main window displays a form with two dropdown menus for material selection, currently showing 'SAC 305' and 'Silicon'. The 'Application Builder' panel on the left shows the project structure, with a red arrow pointing to the 'String' declaration under 'Declarations'. The 'Settings' panel on the right shows the 'List of Variables' table, with a red arrow pointing from the text 'Initial value for strings' to the 'Initial value' column. The 'Errors and Warnings' panel at the bottom shows 0 errors and 0 warnings.

Initial value for strings →

Name	Initial value	Description
var_solder	mat1	String_solder
var_die	mat4	String_die

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”

➤ Constitutive / Materials -> Combo Box



The screenshot shows the COMSOL Multiphysics interface with the following components:

- Application Builder:** A tree view on the left showing the project structure. Under 'Declarations', 'Solder material [choicelist1]' is selected, indicated by a red arrow.
- Materials Choice List:** A central panel showing a list of materials for selection: Solder (SAC 305), Die (Silicon), and another Die (Silicon).
- Settings Panel:** A panel on the right titled 'Settings' for the 'Choice List'. It contains:
 - Choice List:**
 - Label: Solder material
 - Name: choicelist1
 - List Content:** A table defining the items in the choice list:

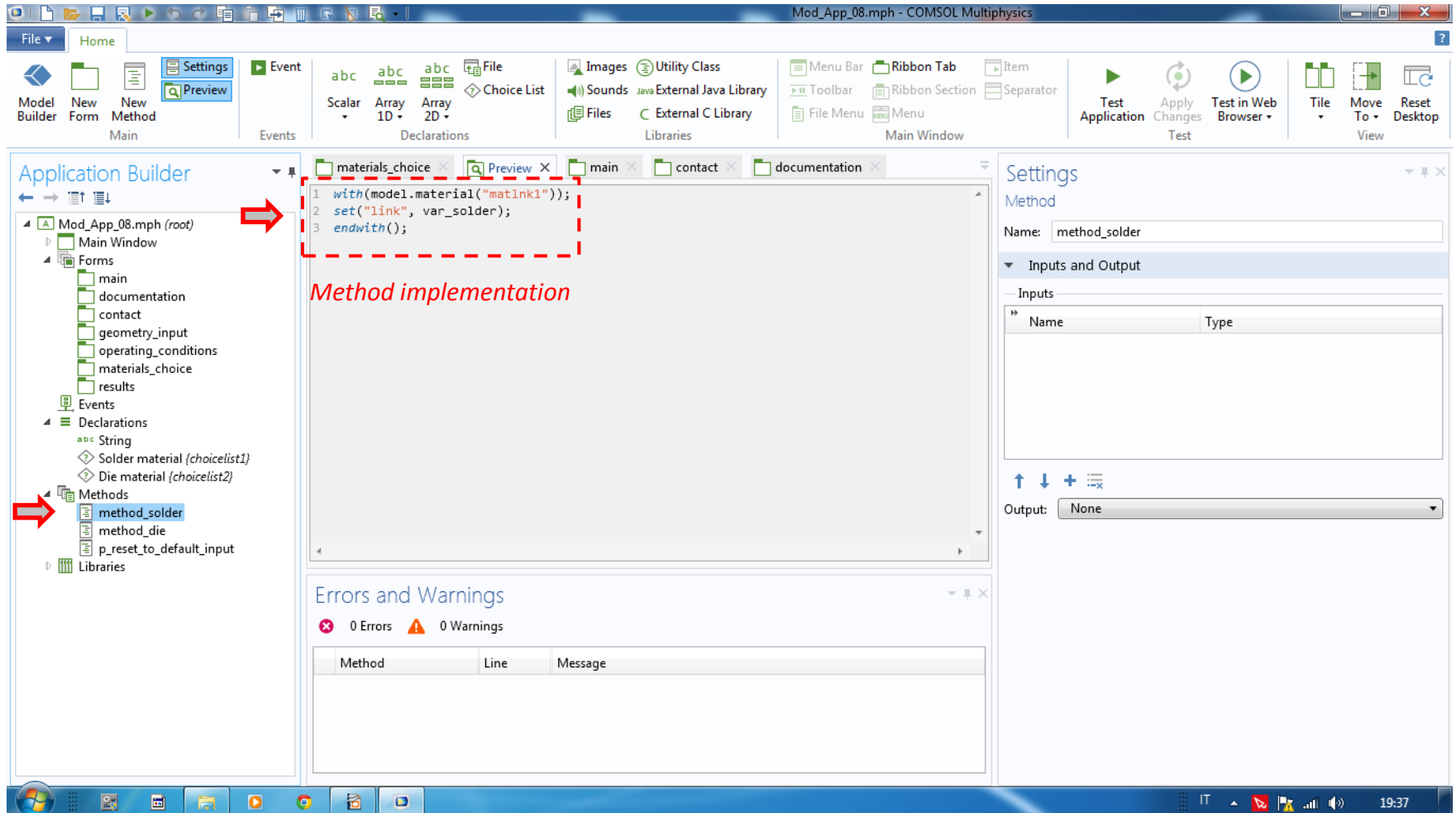
Value	Display name
mat1	SAC 305
mat2	62Sn-36Pb-2Ag
mat3	Ag Sintering
- Errors and Warnings:** A panel at the bottom showing 0 Errors and 0 Warnings.

A red arrow points from the 'Solder material [choicelist1]' declaration in the Application Builder to the 'Settings' panel, and another red arrow points from the 'Settings' panel to the 'List Content' table.

Declaration of choice lists and filling values

Building APP in COMSOL 5.1: "SMD Simulator / Thermal Model"

➤ Constitutive / Materials -> Combo Box



The screenshot displays the COMSOL Multiphysics interface for a project named 'Mod_App_08.mph'. The 'Application Builder' tree on the left shows the project structure, with the 'method_solder' method selected under the 'Methods' folder. A red arrow points from this method to the 'materials_choice' method implementation in the main editor. The implementation code is as follows:

```

1 with(model.material("matlnk1"));
2 set("link", var_solder);
3 endwith();

```

The text *Method implementation* is written in red below the code. The 'Settings' panel on the right shows the configuration for the 'method_solder' method, including its name and input/output settings. The 'Errors and Warnings' panel at the bottom indicates 0 errors and 0 warnings.

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”

➤ Derived values -> Probe / Global variable / Data Display

The screenshot displays the COMSOL Multiphysics interface for a thermal analysis of an SMD (Surface Mount Device). The software window is titled "Mod_App_08.mph - COMSOL Multiphysics".

Model Builder: The left-hand pane shows the model tree. Under "SMD (Surface-Mount Device) (comp1)", the "Definitions" folder is expanded, and "Sonda_Rth (var1)" is selected, indicated by a red arrow.

Properties Panel: The central "Properties" panel is set to "Global Variable Probe". The configuration is as follows:

- Label: Sonda_Rth
- Variable name: var1
- Expression: R_{th} (highlighted with a red dashed box and annotated with "Probe assuming a global variable value")
- Table and plot unit: degC/W

Graphics: The right-hand pane shows a 3D isometric view of the SMD assembly. The model is rendered in a light blue color. Dimensions are visible, including a length of 5 and a width of 1.2. Scale factors of $\times 10^{-3}$ and $\times 10^{-2}$ are indicated on the axes.

Messages: The bottom status bar shows "COMSOL 5.1.0.136" and "Opened file: Mod_App_08.mph".

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”

- Derived values -> Probe / Global variable / Data Display

The screenshot displays the COMSOL Multiphysics interface for a thermal analysis of an SMD (Surface-Mount Device). The software window is titled "Mod_App_08.mph - COMSOL Multiphysics".

Model Builder: The left-hand pane shows the model tree. A red arrow points to the "Variables" node under "Global Definitions". Below it, the "Definitions" folder is expanded, showing "Sonda_T_max (dom1)" and "Sonda_Rth (var1)".

Settings - Properties: The central pane shows the "Variables" settings. A red dashed box highlights the "Variables" table, which contains the following data:

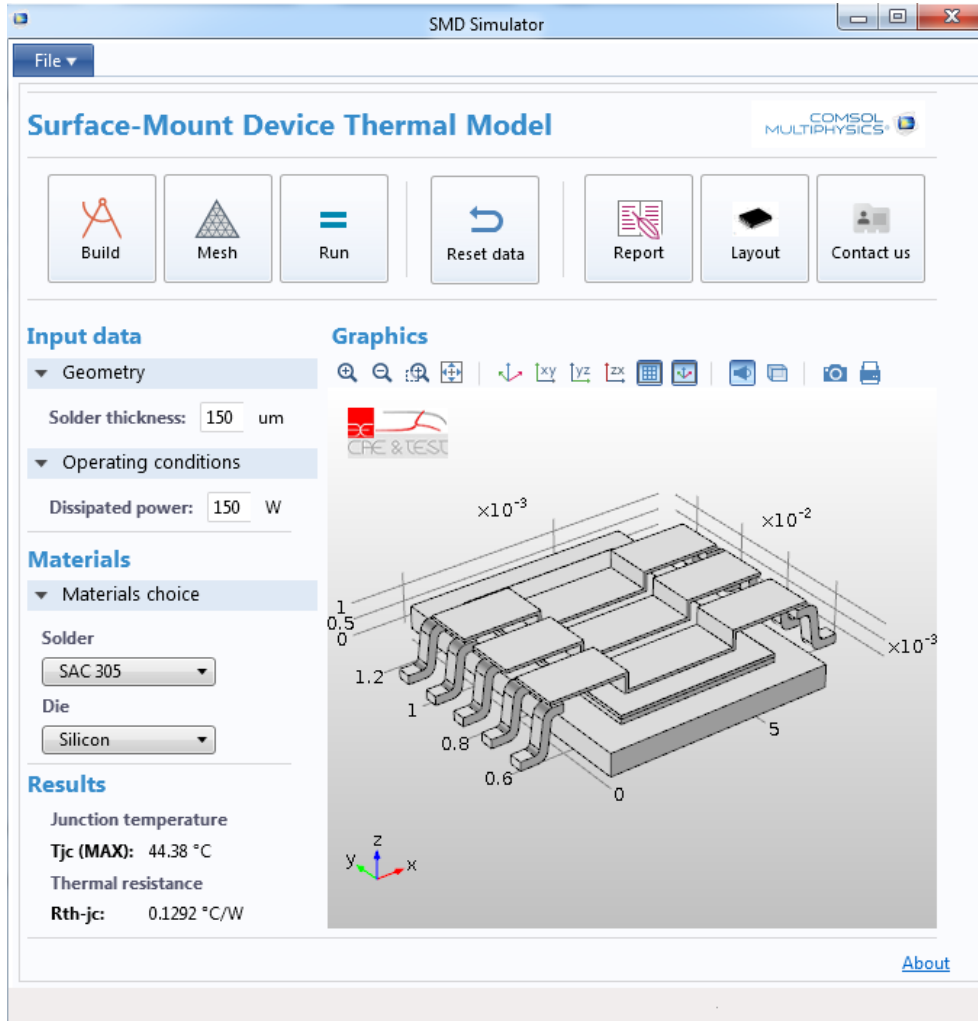
Name	Expression	Unit	Description
Rth	$(\text{root.comp1.dom1-T_ref})/P_max$	K/W	

Below the table, a red text annotation reads: "Variable value depends in this case by another probe value".

Graphics: The right-hand pane shows a 3D isometric view of the SMD component. The model is rendered in a light gray color. The coordinate system (x, y, z) is visible at the bottom left. The dimensions of the component are indicated by blue dimension lines: 1.2, 0.8, 0.6, 1, 0.5, 0, 1, 5, and 0. The scale factors $\times 10^{-3}$ and $\times 10^{-2}$ are shown near the dimensions.

Messages: The bottom status bar shows the following information: "COMSOL 5.1.0.136", "Opened file: Mod_App_08.mph", and "874 MB | 1076 MB".

Building APP in COMSOL 5.1: “SMD Simulator / Thermal Model”



The screenshot displays the COMSOL SMD Simulator interface. The title bar reads "SMD Simulator". The main window is titled "Surface-Mount Device Thermal Model".

Input data:

- Geometry: Solder thickness: 150 μm
- Operating conditions: Dissipated power: 150 W

Materials:

- Solder: SAC 305
- Die: Silicon

Results:

- Junction temperature
- $T_{jc} \text{ (MAX): } 44.38 \text{ } ^\circ\text{C}$
- Thermal resistance
- $R_{th-jc}: 0.1292 \text{ } ^\circ\text{C/W}$

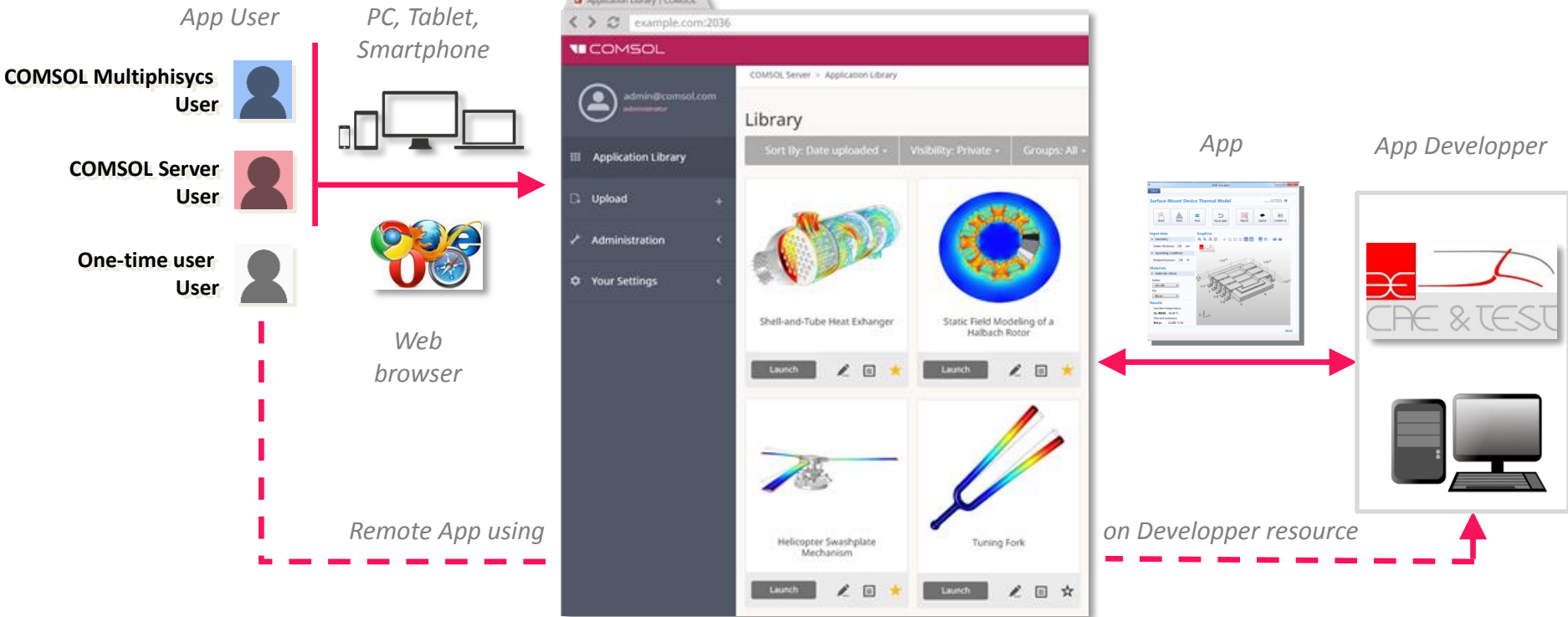
The 3D model shows a die with dimensions 1.2 $\times 10^{-3}$ m by 0.8 $\times 10^{-3}$ m, mounted on a substrate with a 5 $\times 10^{-2}$ m wide pad. The solder thickness is 150 $\times 10^{-6}$ m. A coordinate system (x, y, z) is shown at the bottom left of the model.

➤ How to use it?

Launching APP in COMSOL SERVER



App platform: COMSOL Server





BE CAE & TEST S.r.l.

Viale Africa 170, Sc.A
95129 Catania (Italy)

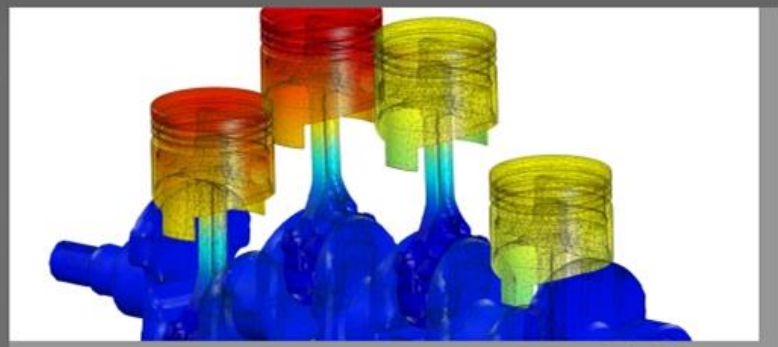
Phone +39 095 216 64 26

URL: <http://www.be-caetest.it>

E-mail: info@be-caetest.it

PEC: info@pec.be-caetest.it

- Activities
- Services
- Partners
- News
- Contact us



CAE & Test

Our team of engineers and researchers supports companies and individuals during their products and processes development by using advanced CAD/CAE tools and organizing accurate experimental tests.

The experience gained in experimental vibro-acoustics, FEM and Multibody numerical simulation and the use of Multiphysics packages make BE CAE & Test the ideal partner to guarantee reliability, innovation and competitiveness of your products and processes.

© 2015 BE CAE & Test S.r.l. | P.I. 05230570870

<https://www.comsol.it/certified-consultants/bus>