

BE CAE & Test



An Application Built with the COMSOL Multiphysics® Software for Managing Computational Sequences in Thermal Fluid Applications

Giuseppe Petrone and Carmelo Barbagallo

BE CAE & TEST, Viale Africa 170 Sc. A, 95129 Catania (ITALY)

Munich, 12-14 October 2016



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



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/>

COMSOL
Certified Consultant

COMSOL

Prodotti Workshop Webinar Supporto Contatti

BE CAE & Test

BE CAE & Test provides consultancy services for applications based on numerical simulations. Our company recognizes that virtual prototyping plays an important role in several engineering fields. It serves as a powerful tool for optimizing the design of products and processes, while also reducing the time to market ratio. We strongly believe that complex problems have to be analyzed using a multiphysics approach.

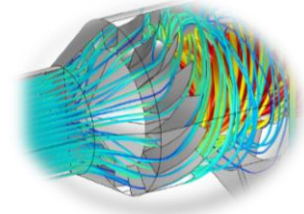
Our team of engineers and researchers supports companies and individuals during their products and processes development by using advanced CAD/CAE tools and organizing accurate experimental tests.

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

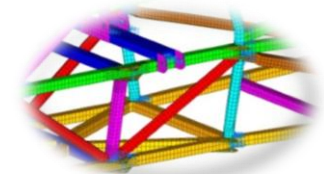
➤ **Fluid dynamics and thermal analyses**

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



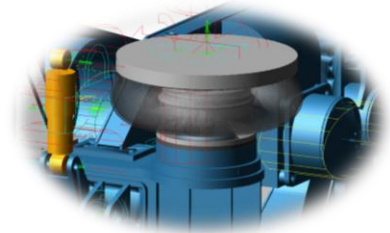
➤ **Structural analyses**

- Linear and non-linear statics, dynamic and vibro-acoustics 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

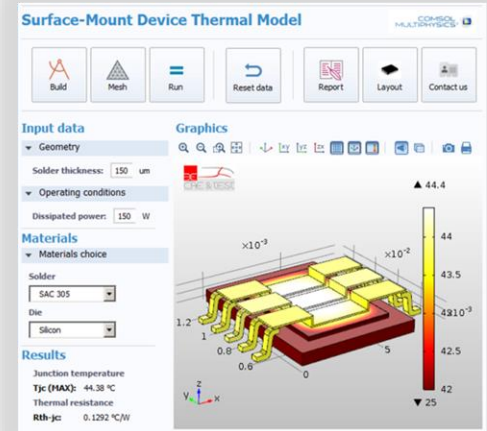


➤ **Experimental testing**

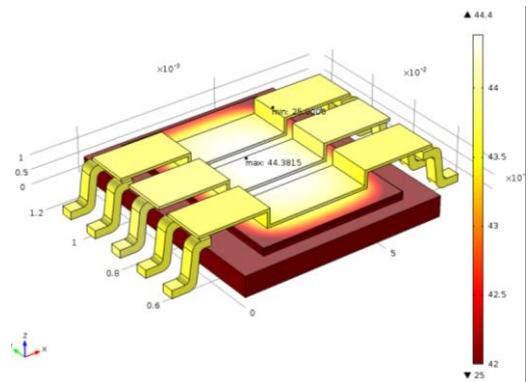
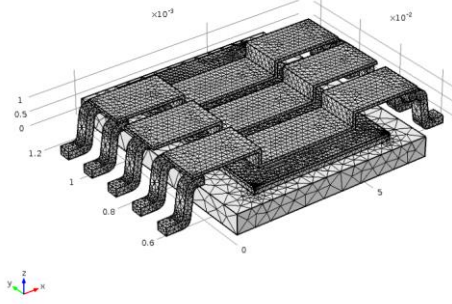
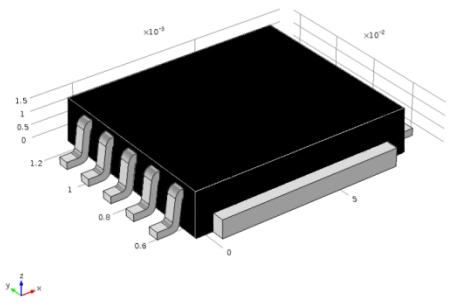
- Ride comfort (NVH), modal analyses
- Vibro-acoustics



➤ **BE CAE & Test strongly supports the concept «FROM MODEL TO APP»**



FROM MODEL TO APP

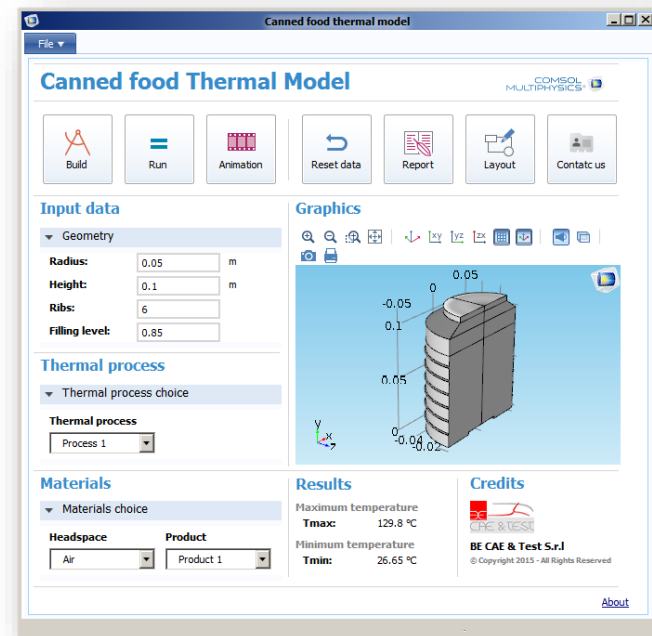


➤ Why building a COMSOL App*?

*“... Apps mark a **revolutionary page** in the history of mathematical **modeling** and numerical simulation: these specialized and **user-friendly tools bring the power of numerical simulation to a larger group of users...***”

“... people with no prior experience from FEA or mathematical modeling can access, exploit, and benefit from analysis...”

*“...simulation apps can create **more business opportunities with customers**. Beyond simply providing them with a technical report, you are also supplying them with an interactive tool...they can use to **investigate the problem on their own ...**”*



*Extract from COMSOL Press Release «Simulation Apps Pave New Frontier for Virtual Prototyping of Surface-Mount Devices»
<https://www.comsol.com/press/news/article/3231/>

➤ Why building a COMSOL App*?

*"... Apps mark a **revolutionary page** in the history of mathematical **modeling** and numerical simulation: these specialized and **user-friendly tools** bring the **power of numerical simulation** to a **large group of users**..."*

*"... people with no prior experience from FEA or mathematical modeling can **access, exploit, and benefit from analysis**..."*

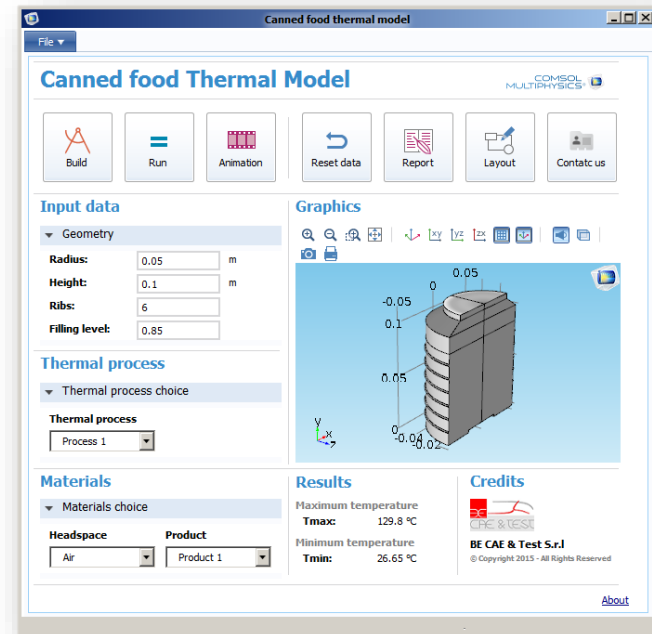
*"...simulation apps can create **more business opportunities with customers**. Beyond simply providing them with a technical report, you are also supplying them with an interactive tool...they can use to **investigate the problem on their own**..."*

FOCUS ON CUSTOMER



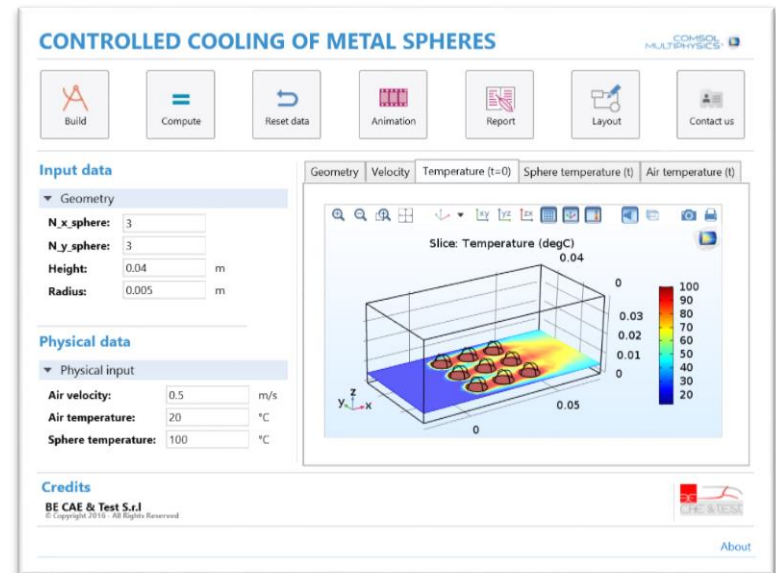
Can the model developer have a benefit for himself by using the Application Builder?

- Try to use the Application Builder to make our work easier
- Automating models by using the Application Builder



- We built a demo Comsol App to investigate this opportunity:

An Application Built with the COMSOL Multiphysics® Software for Managing Computational Sequences in Thermal Fluid Applications



Can the model developer have a benefit for himself by using the Application Builder?

- Try to use the Application Builder to make our work easier
- Automating models by using the Application Builder

HEAT TREATING

Heat treating is a group of **industrial processes** used to **alter** the physical **properties of a material**. The most common application is **metallurgical**.

Heat treatment **involves** the **use of heating or chilling**, normally to extreme temperatures, to achieve a desired result such as hardening or softening of a material.

Thermal management is strongly required.



MODELLING & SIMULATION

- Optimizing the heating/cooling equipment.
- Monitoring the time evolution of the pieces “core” temperature as a function of :
 - Size;
 - Constituting material;
 - Relative position of the pieces in the cooling equipment;
 - Magnitude of the cooling forced flow.;
 -



HEAT TREATING

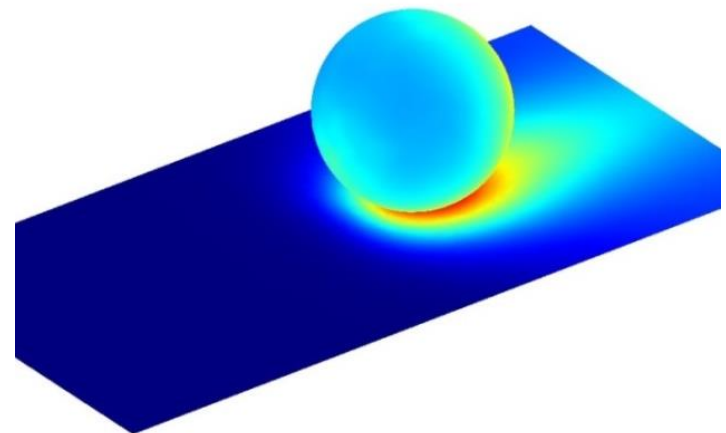
Heat treating is a group of **industrial processes** used to **alter** the physical **properties of a material**. The most common application is **metallurgical**.

Heat treatment **involves** the **use of heating or chilling**, normally to extreme temperatures, to achieve a desired result such as **hardening or softening** of a material.



MODELLING & SIMULATION

Very usually, it is required to **simulate a transient thermal analysis** in a **forced permanent flow** and starting from an **initial thermal state**.



HEAT TREATING

Heat treating is a group of **industrial processes** used to **alter** the physical **properties of a material**. The most common application is **metallurgical**.

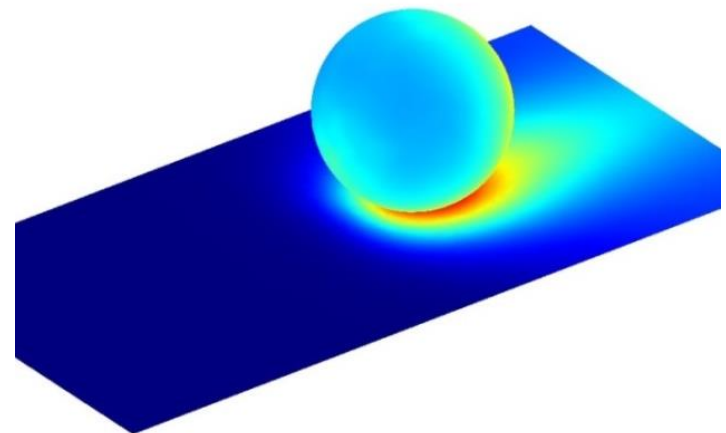
Heat treatment **involves** the **use of heating or chilling**, normally to extreme temperatures, to achieve a desired result such as **hardening or softening** of a material.



MODELLING & SIMULATION

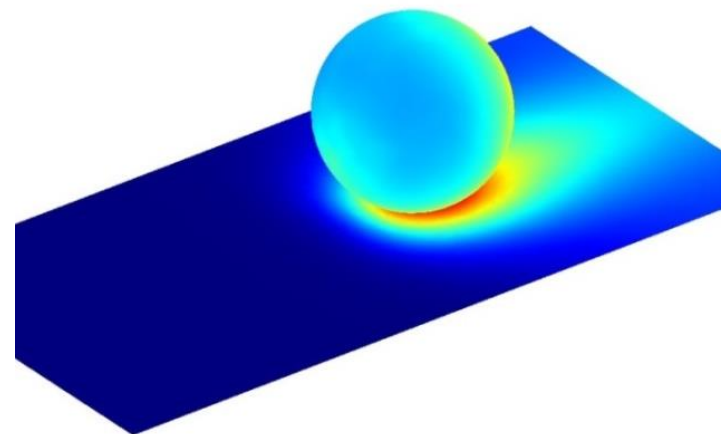
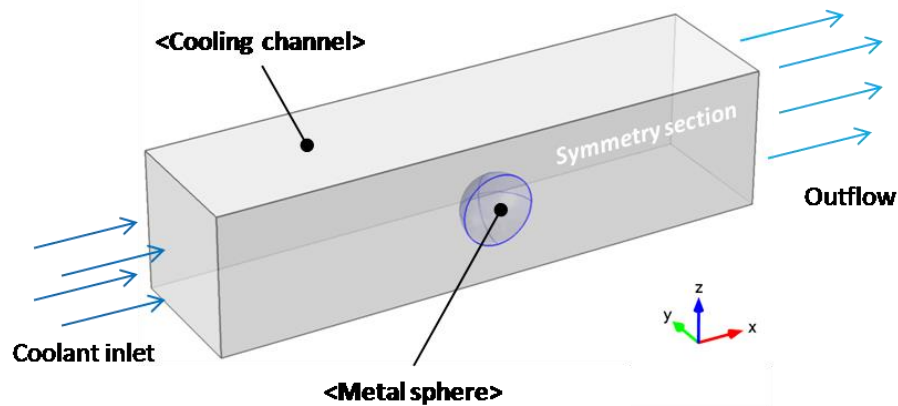
This kind of study can be carried out by **three steps**:

1. **Fluid dynamical** solution of the **permanent velocity field**;
2. **Steady thermal simulation** of temperature at the begin of the cooling process;
3. **Transient thermal analysis** during the cooling process.



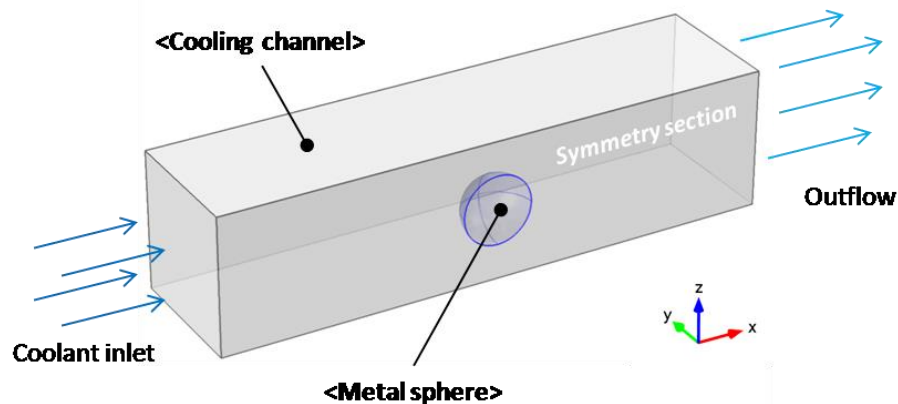
Assumptions / constraints / modelling steps

- Solving the **steady fluid dynamical** flow
- Set a chosen **temperature** value at the solid-fluid interface of the cooled objects to get consistent initial thermal state for the process
- Solving the **thermal steady state** by using the velocity field as transport field
- Set a a continuity thermal condition at the solid-fluid interface
- Solving the **transient analysis** using the velocity field as transport field and the thermal steady state as initial condition

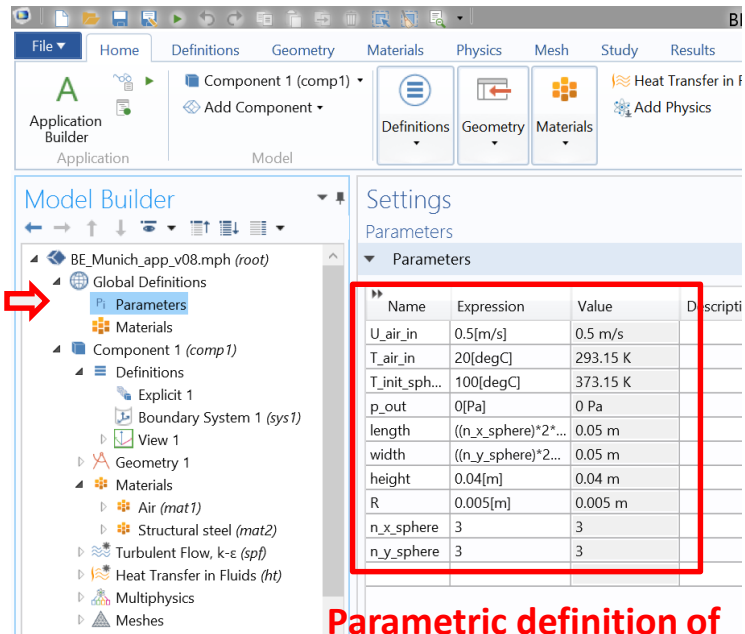


Assumptions / constraints / modelling steps

- Solving the **steady fluid dynamical** flow
- Set a chosen **temperature** value at the solid-fluid interface of the cooled objects to get consistent initial thermal state for the process
- Solving the **thermal steady state** by using the velocity field as transport field
- Set a a continuity thermal condition at the solid-fluid interface
- Solving the **transient analysis** using the velocity field as transport field and the thermal steady state as initial condition



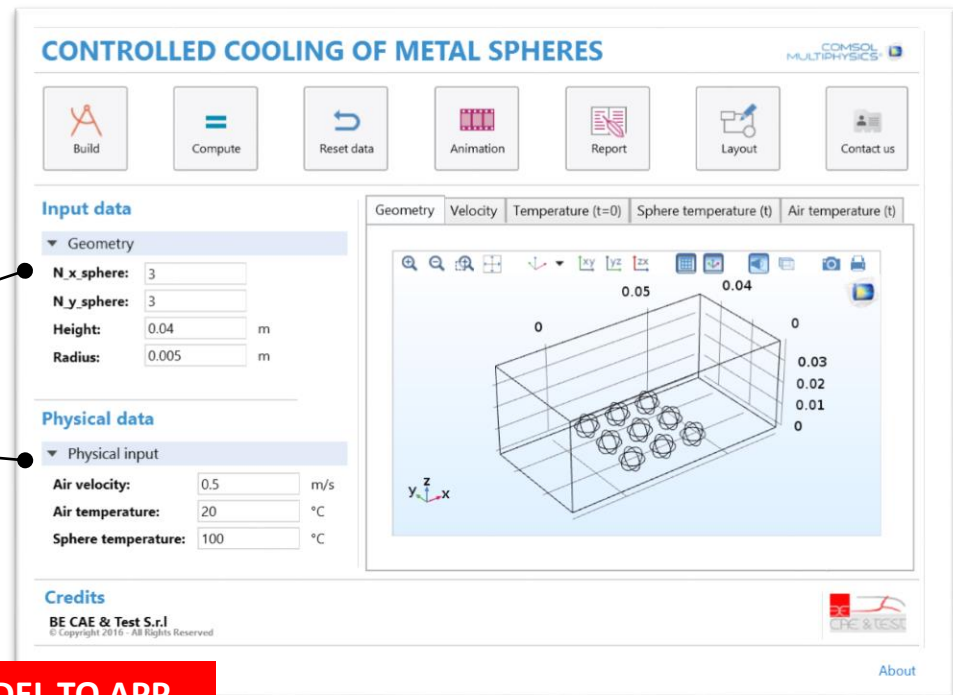
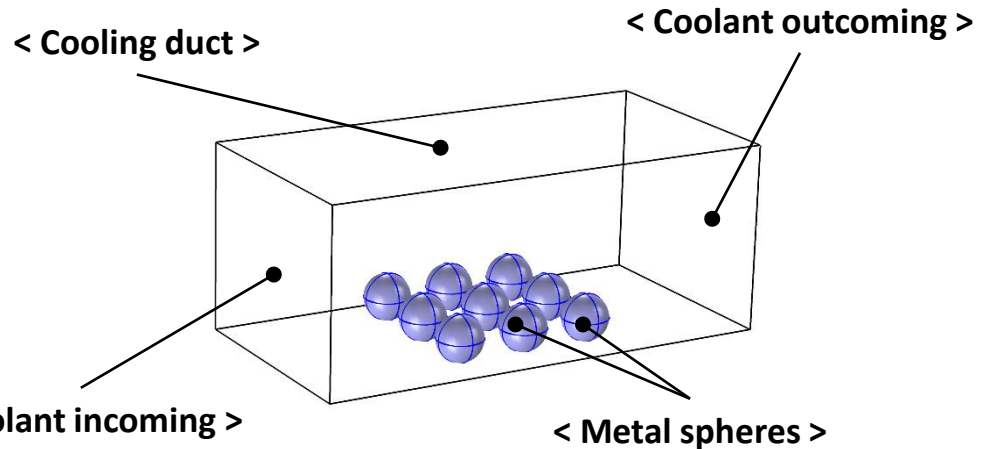
CAN A COMSOL APP HELP US TO AUTOMATICALLY MANAGE THE SEQUENCE OF SEVERAL COMPUTATIONALS STEPS TO BE RUN BY USING DIFFERENT PHYSICAL AND SOLVING SETTINGS ?

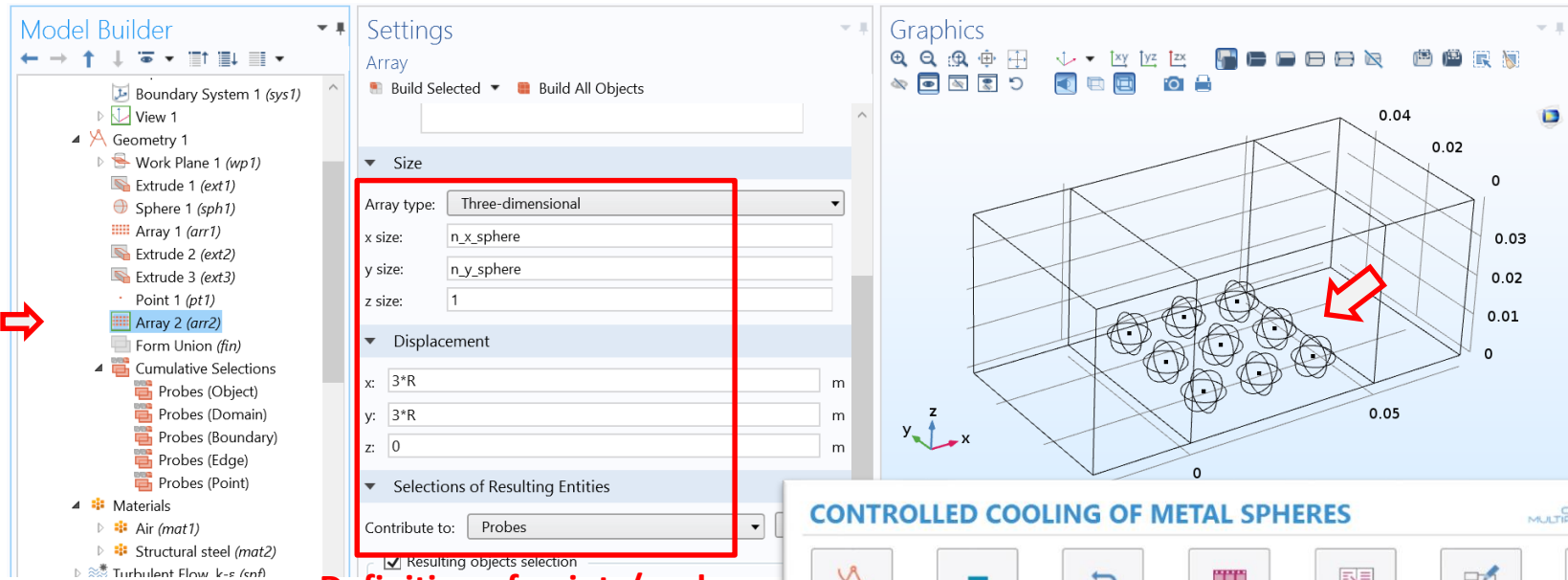


Parametric definition of geometric and process input

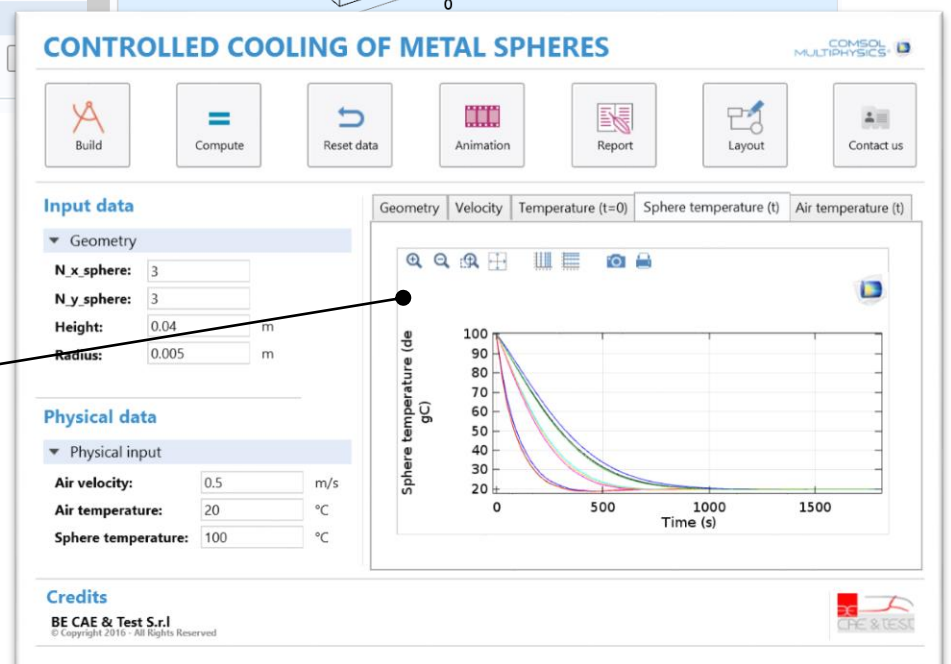
Parametric set-up

- **Geometrical** (spheres number and dimension, duct dimensions)
- **Functional** (Coolant flowrate and temperature, initial spheres temperature)



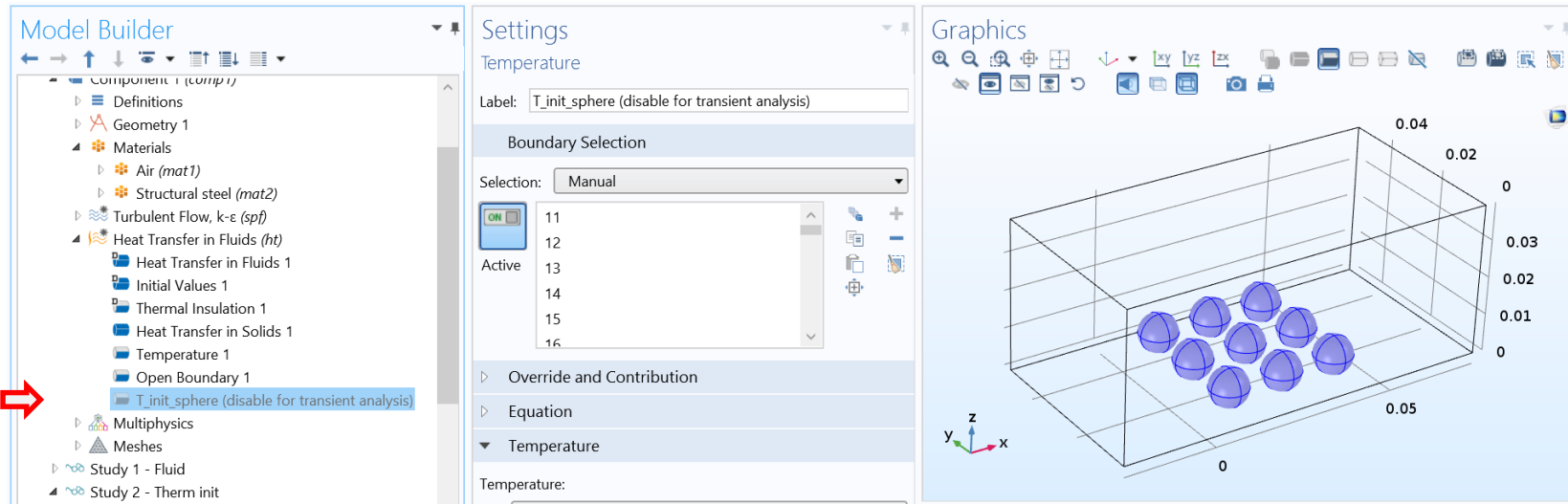


**Definition of points/probes
(parametric also)**



Parametric set-up

- **Derived values** (time-evolution of the «core» temperature for each sphere, time-evolution of the average temperature of out-flowing air)



The screenshot displays three panels from a software application:

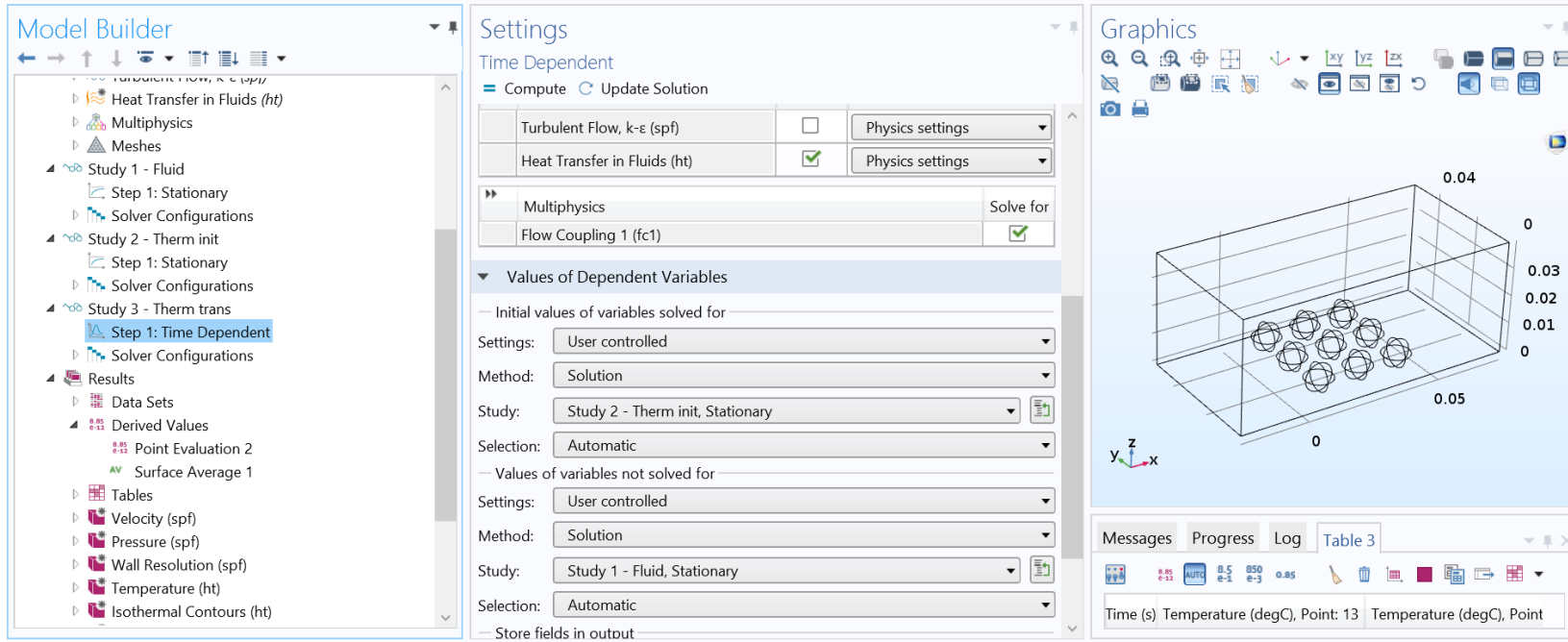
- Model Builder:** A tree view on the left showing a hierarchy of components. A red arrow points to the item `T_init_sphere (disable for transient analysis)` under the `Temperature 1` folder.
- Settings:** A central panel titled "Temperature" with a "Boundary Selection" section. It includes a "Selection" dropdown set to "Manual" and a list of active boundaries (11-16). Below this are sections for "Override and Contribution", "Equation", and "Temperature".
- Graphics:** A 3D visualization on the right showing a rectangular domain with a grid. Inside the domain, several blue spheres are arranged in a cluster. The axes are labeled x, y, and z, with numerical values ranging from 0 to 0.05.

Physics setting
 (Initial temperature at solid-fluid interface)

The screenshot displays the ANSYS Workbench interface. On the left is the **Model Builder** tree, showing a sequence of studies: Study 1 - Fluid, Study 2 - Therm init, and Study 3 - Therm trans. The **Settings** panel in the center is set to **Time Dependent** and shows a list of physics: Turbulent Flow, k-ε (spf), Heat Transfer in Fluids (ht), Multiphysics, and Flow Coupling 1 (fc1). A red box highlights the **Values of Dependent Variables** section, which is used to define the coupling between studies. Two red arrows point to the **Study** dropdown menus, one for 'Study 2 - Therm init, Stationary' and another for 'Study 1 - Fluid, Stationary'. The **Graphics** panel on the right shows a 3D model of a rectangular box with several spherical objects inside, with axes labeled x, y, and z.

Study setting

(A sequence of studies has to be defined by coupling variable values from a study to another)



«Manual» work-flow by using the embedded model:

- Run < Study 1 – Fluid > for solving the velocity and pressure fields
- Set (enable) specific BCs (surface spheres temperature)
- Run < Study 2 – Therm init > to get the specific thermal state at the initial time
- Disable specific BCs (surface spheres temperature)
- Run < Study 3 – Therm trans > to perform the transient analysis

**Waiting for
solution**

**Waiting for
solution**

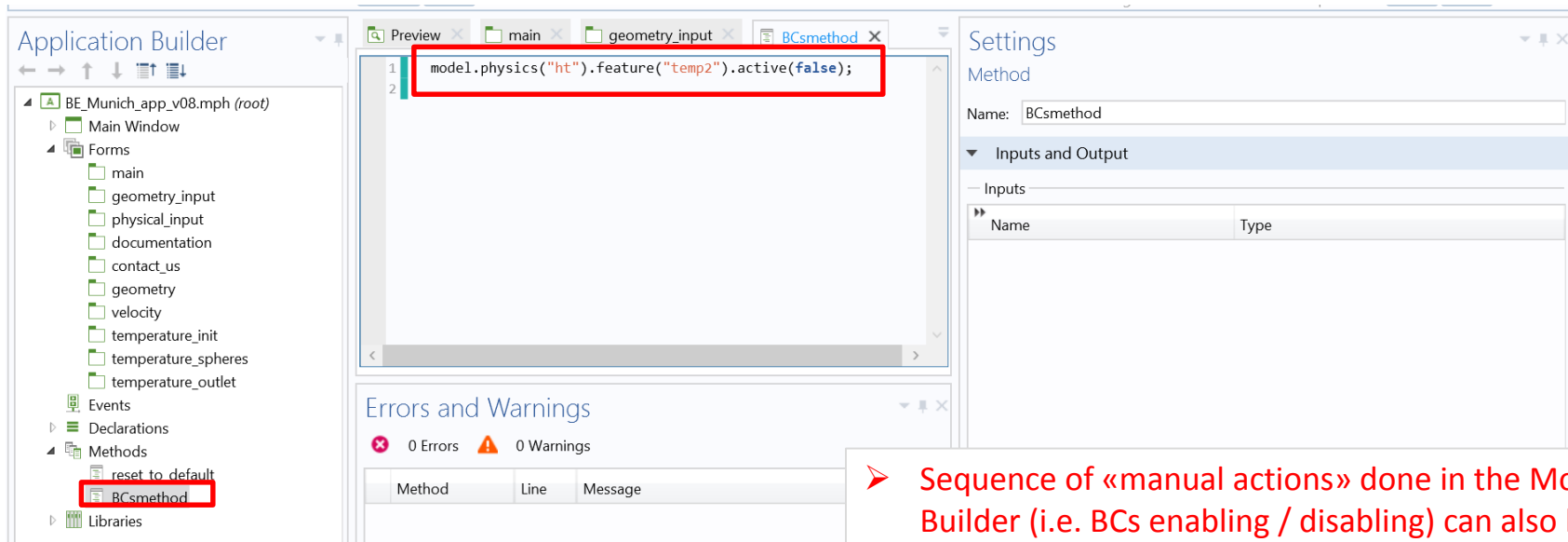
➤ The Application Builder allows to EXECUTE a sequence of commands by a single «action button» ...

«Manual» work flow by using the embedded model:

- Run < Study 1 – Fluid > for solving the velocity and pressure fields
- Set (enable) specific BCs (surface spheres temperature)
- Run < Study 2 – Therm init > to get the specific thermal state at the initial time
- Disable specific BCs (surface spheres temperature)
- Run < Study 3 – Therm trans > to perform the transient analysis

Waiting for solution

Waiting for solution



The screenshot displays the 'Application Builder' software interface. On the left, a project tree shows a hierarchy starting with 'BE_Munich_app_v08.mph (root)', followed by 'Main Window', 'Forms', and 'main'. Under 'main', there are several sub-items including 'geometry_input', 'physical_input', 'documentation', 'contact_us', 'geometry', 'velocity', 'temperature_init', 'temperature_spheres', and 'temperature_outlet'. The 'Methods' folder is expanded, showing 'reset to default' and 'BCsmethod', with 'BCsmethod' highlighted by a red box. The central code editor shows two lines of code, with the second line, `model.physics("ht").feature("temp2").active(false);`, highlighted by a red box. To the right, the 'Settings' panel is open, showing the 'Method' settings for 'BCsmethod'. Below the 'Inputs and Output' section, there is a table with columns for 'Name' and 'Type'. At the bottom, the 'Errors and Warnings' panel shows '0 Errors' and '0 Warnings'.

➤ Sequence of «manual actions» done in the Model Builder (i.e. BCs enabling / disabling) can also be recorded: the available code can then be «applied» to an action button, in order to automatically reply the sequence of commands ...

«Manual» work flow by using the embedded model:

- Run < Study 1 – Fluid > for solving the velocity and pressure fields
- Set (enable) specific BCs (surface spheres temperature)
- Run < Study 2 – Therm init > to get the specific thermal state at the initial time
- Disable specific BCs (surface spheres temperature)
- Run < Study 3 – Therm trans > to perform the transient analysis

Waiting for solution

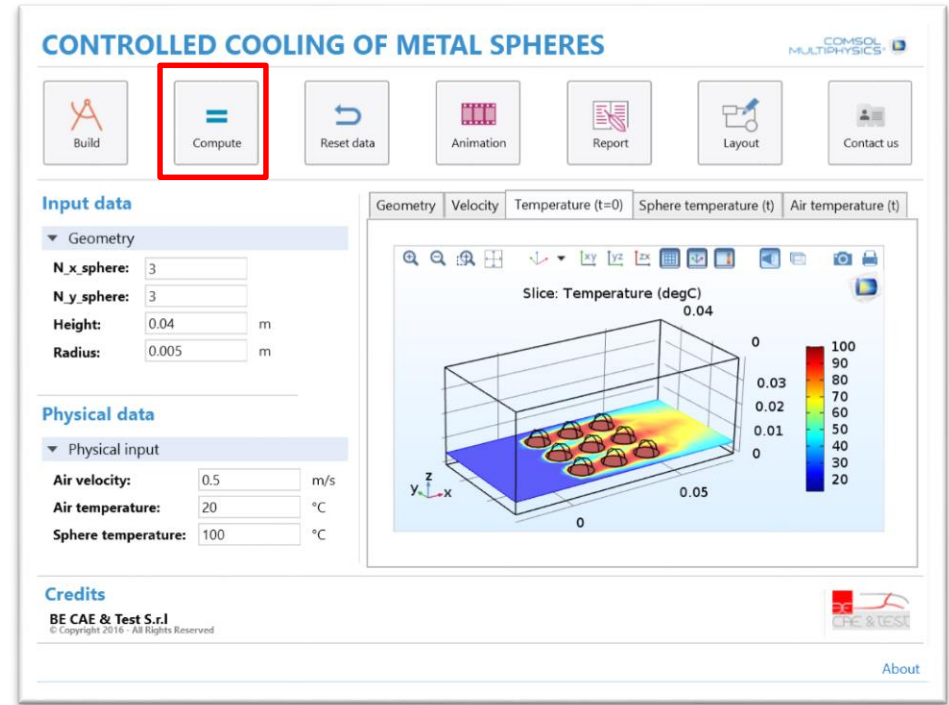
Waiting for solution

«Manual» work flow by using the App:

- Click the button «Compute» ...



FROM MODEL TO APP



«Manual» work flow by using the embedded model:

- Run < Study 1 – Fluid > for solving the velocity and pressure fields
- Set (enable) specific BCs (surface spheres temperature)
- Run < Study 2 – Therm init > to get the specific thermal state at the initial time
- Disable specific BCs (surface spheres temperature)
- Run < Study 3 – Therm trans > to perform the transient analysis

Waiting for solution

Waiting for solution



BE CAE & TEST S.r.l.

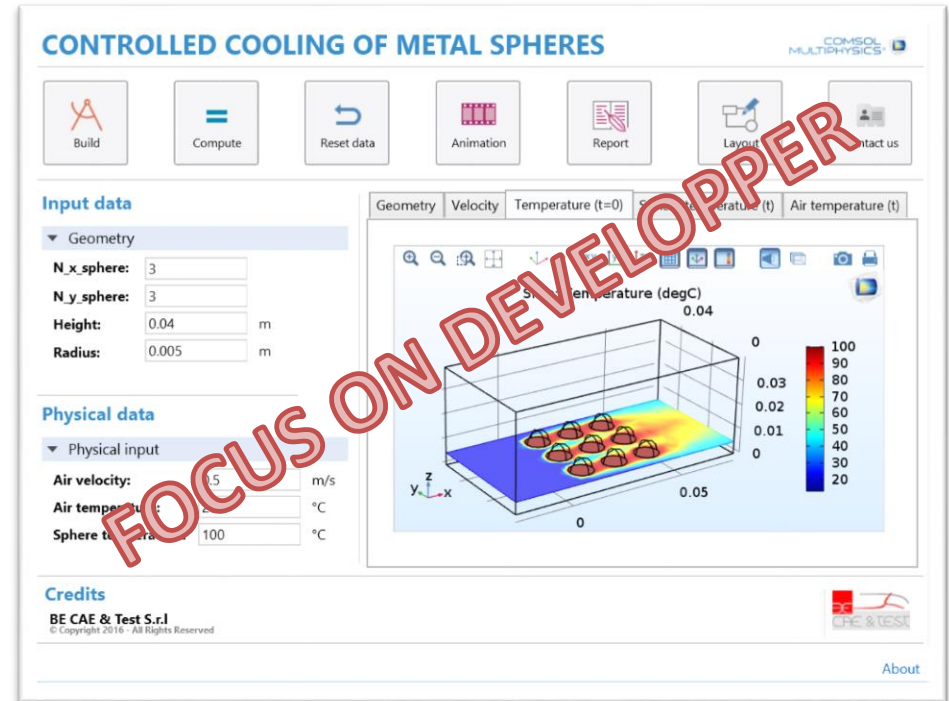
Viale Africa 170, Sc.A
95129 Catania (Italy)

Phone +39 095 216 64 26

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

E-mail: info@be-caetest.it

PEC: info@pec.be-caetest.it



THANK YOU!

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