Humidity evolution (breathing effect) in enclosures with electronics

Hygum, Morten Arnfeldt; Popok, Vladimir

Published in:
IMAPS Nordic Annual Conference 2015 June 8-9, Helsingør

Publication date:
2015

Document Version
Accepted author manuscript, peer reviewed version

Humidity evolution (breathing effect) in enclosures with electronics

Morten A. Hygum and Vladimir N. Popok

Aalborg University, Skjernvej 4a, 9220 Aalborg East, Denmark

Phone:+45 9940 9215, Fax:+45 9940 9235 and E-mail: mah@nano.aau.dk

Abstract

Packaging and enclosures used for protecting power electronics operating outdoors are designed to withstand the local climatic and environmental changes. Hermetic enclosures are expensive and therefore other solutions for protecting the electronics from a harsh environment are required. One of the dangerous parameters is high humidity of air. Moisture can inevitable reach the electronics either due to diffusion through the wall of an enclosure or small holes, which are designed for electrical or other connections. A driving force for humid air movement is the temperature difference between the operating electronics and the surrounding environment. This temperature, thus, gives rise to a natural convection, which we also refer to as breathing. Robust and intelligent enclosure designs must account for this breathing as it can significantly change the humidity distribution in the enclosure.

In the current work we suggest a modelling procedure to investigate a breathing effect for an enclosure with opening (hole). The simulations are carried out by solving an energy equation coupled with the Navier-Stokes equation. The movement of moisture is considered through a convection-diffusion equation. The approach is verified by measuring the temperature and humidity profiles in a test setup (container) while also considering the moisture flux outside the container. The test setup is a vertical cylinder enabling to simplify the modeling to 2D case. The experimental measurements are compared to simulations and good agreement is obtained.

Key words: Modeling of humidity distribution, Enclosures with electronics.

1 Introduction

In reliability engineering a significant issue is the problem of climatic simulations which includes the aspect of relative humidity (RH) of air. It is a well-known fact that a humid climate greatly affects the lifetime of electronics.

Completely hermetic boxes or cabinets are quite expensive and therefore not used in routine packaging technology. The humidity and temperature inside a cabinet are thus affected by the climate outside the packaging e.g. through small openings. Operating electronics inevitably heats the environment inside an enclosure in which it is installed. Thus, the packaging will be exposed to thermal gradients. Due to gravity, this leads to pressure gradients in the air inside the enclosure that gives rise to movement of the air. This natural convection, sometimes also referred to as breathing, can transport water vapor from/in the enclosure. In order to design not expensive and intelligent packaging solutions which ensure good reliability towards high humidity, simulations based on the physics-based climatic models of such systems are of importance.

To our knowledge, a significant attention has been paid to predict the moisture absorption by the electronic packaging, the moisture distribution inside the encapsulating material and at circuit boards, see e.g. [1, 2, 3, 4, 5]. However, approaches to predict the climatic conditions, to which the electronics is exposed to, have not been intensively studied. Here, an investigation of the water vapor mass flow through openings in a test setup (enclosure) is presented. The experiments are designed to obtain the values of the mass flow through the openings of different diameters.

2 Theory

Theoretical prediction of natural convection is a complicated matter because it involves coupling the Navier-Stokes equations with the energy equation using Boussinesq approximation. The generated pressure gradient is considered with a buoyancy force and the air is set to be incompressible [7]. The momentum conservation equation is

$$\rho\left(\frac{\partial \vec{u}}{\partial t} + (\nabla\vec{u})\cdot\vec{u}\right) = -\nabla p + \vec{\nabla}\cdot\tau + \vec{f}, \qquad (1)$$

where ρ is the density, t is the time, p is the pressure, $\vec{f}$ is the body force and the viscous stress tensor is

$$\tau = \nu(\nabla\vec{u} + (\nabla\vec{u})^T) - \frac{2}{3}\nu(\nabla\vec{u})\boldsymbol{I}, \qquad (2)$$

with $\boldsymbol{I}$ to be the identity matrix and ν to be the dynamic viscosity. The velocity field $\vec{u}$ obeys the continuity equation for incompressible fluids

$$\frac{\partial \rho}{\partial t} + \rho \vec{\nabla} \cdot \vec{u} = 0. \quad (3)$$

Equation (1) and (2) are the Navier-Stokes equations. The continuity equation for the energy is

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \vec{u} \cdot \nabla T = \nabla \cdot (k \nabla T) + Q, \quad (4)$$

with C_p to be the heat capacity at constant pressure, T to be the temperature, k to be the heat conductivity and Q to be a source term. The coupling between equation (1) and (3) is through the body force term $\vec{f}$. The Boussinesq approximation enables:

$$\vec{f} = \rho \vec{g} = \left(\bar{\rho} - \bar{\rho}\bar{\beta}(T - \bar{T})\right)\vec{g}\,, \quad (5)$$

where β is the thermal expansion coefficient and $\vec{g}$ is the acceleration due to gravity. In this study these non-linear, coupled partial differential equations must be solved numerically. By simultaneously solving of the equations the temperature and movement of the air inside the enclosure is obtained. For reliability engineering the amount of water vapor in the air is the most critical parameter. The transport of water vapor can be modeled as movement of a diluted species in air. This is likewise governed by the continuity equation:

$$\frac{\partial c}{\partial t} + \vec{\nabla} \cdot \vec{J} = 0, \quad (6)$$

where c is the concentration of water vapor and $\vec{J}$ is the vapor flux, which is divided into a diffusion part and a convective part, as follows:

$$\vec{J} = -D_C \vec{\nabla} c + c\vec{u}, \quad (7)$$

with D_c to be the diffusion coefficient [6, 7]. The vapor pressure is given as

$$p_v = cRT, \quad (8)$$

with R as the ideal gas constant. The vapor pressure allows the RH obtained from

$$RH = \frac{p_v}{p_{sat}}, \quad (9)$$

where p_{sat} is the saturated vapor pressure.

2 Experiments

The test setup is made in shape of a cylinder that allows to reduce the model to a 2D axissymmetric domain. A cylindrical wetted sponge was then inserted into the middle of the cylinder with a steel wire, where the other end of the steel wire was attached to a scale which was placed over the test setup. At the top of the tube an end cap was mounted with a small hole in the middle. The hole size was varied for each experimental series. At the bottom of the tube a Peltier element was mounted. A schematic of the setup is shown in figure 1.

The experiments were conducted by lowering the wetted sponge into the test setup, attaching the other end of the steel wire to a scale above the setup, covering the opening with a napkin to avoid convective disturbance from the surroundings. Then, one should wait until the air inside the tube was saturated with water vapor and weigh the sponge. The next stage was to heat the setup using the Peltier element. After the heating procedure was done the sponge was weighed again and an average flux of water vapor through the opening could be calculated from the known mass difference of the sponge and evaporation time. The starting temperature of the experiments was 21°C.

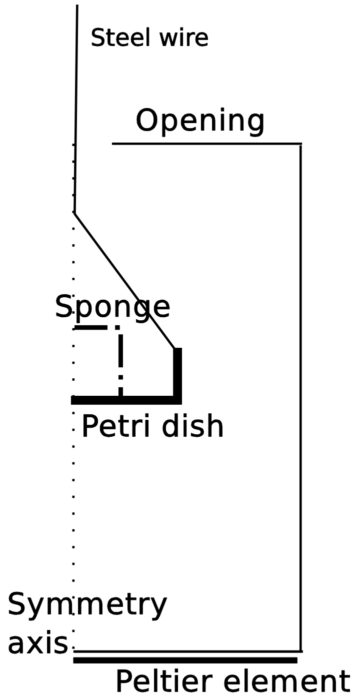

Figure 1: 2D schematic of the half of the cylindrical setup.

To obtain significantly weight losses from the sponge the heating profile was applied 3-5 times in a row depending on the size of the opening. To isolate sources of error, two different heating profiles were tested. The first profile is shown in figure 2 while the second profile is shown in figure 3. The first heating profile was carried out by heating the device to 29°C and letting the device cool. This was done to mimic an operating circuit board. Reasons for the second profiles are mentioned in detail in section 5.

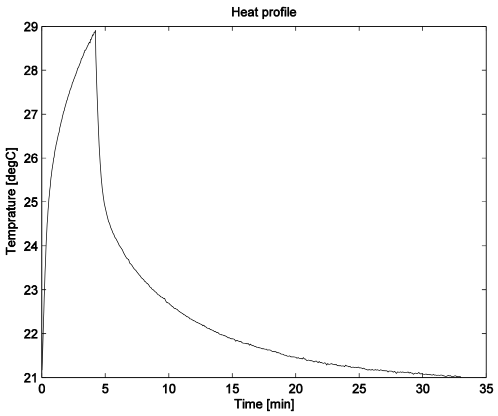

Figure 2: The first heating profile as a function of time.

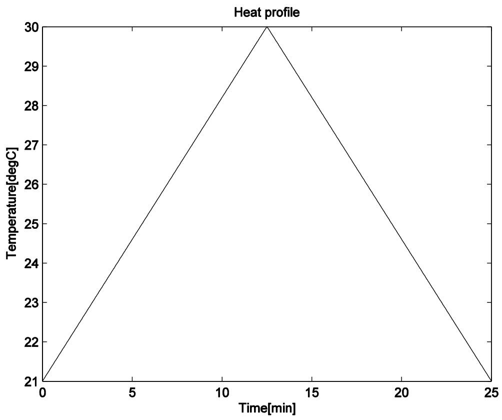

Figure 3: The second heating profile as a function of time.

4 Modeling

The modelling was carried out with COMSOL Multiphysics. All material properties are that of humid air and obtained from [8].

The heating profile was loaded into the software and applied as a boundary condition at the bottom of the tube. Otherwise all boundary conditions and initial conditions were set to $T = 21°\mathrm{C}$.

The initial condition was set to still air. All the walls are set to no-slip conditions. At the opening the pressure was set to atmospheric value.

Throughout the chamber, the no-flux boundary condition was used, except for the opening and the sponge. At the sponge, the water vapor concentration was set to that of a RH of 100 %, as follows:

$$c_{sponge} = \frac{p_{sat}}{RT}, \qquad (10)$$

where the saturated vapor pressure p_{sat} for water vapor is given as [9]

$$p_{sat}(T) = 2.53 \cdot 10^{11} e^{-\frac{2.501\cdot\frac{10^6}{461.5}}{T}} [Pa]. \qquad (11)$$

At the opening the following boundary condition was applied:

$$c_{opening} = \frac{p_{sat}}{2RT}, \qquad (12)$$

which amounts to the water vapor concentration of a RH of 50 %. The initial condition was the vapor concentration corresponding to a RH of 90 %.

$$c_{initial} = 0.9\frac{p_{sat}}{RT}, \qquad (13)$$

The following diffusion coefficient was used:

$$D_{H_2O,air} = 1.87 \cdot 10^{-6} \cdot T^{2.072} \left[\frac{m^2}{s}\right], \qquad (14)$$

which is valid for the temperature range 282-450K [10].

5 Results and discussion

The modeled and measured average dissipation of vapor for the first heating profile are shown in table 1. Since the values are small we consider the agreement to be satisfactory (same order of magnitude) for small openings. The larger difference between the model and experiments for the opening with radius 2 cm is believed to be due to convective disturbances of the test setup surrounding Movement of air at the top of the cylinder would mix the air from outside the setup with the air inside it. This would lead to larger concentration gradients in water vapor which causes large diffusion flux of water vapor out of the setup. Another possible error, which is worth mentioning, is that the model being done with the Boussinesq approximation itself leading to not very appropriate simulation of the air compression inside the cylinder. Thus, the deviations in the heating profile can cause a difference between the model and experiments.

Table 1. *Calculated and measured dissipation of vapor for heat profile in the first series of experiments.*

OPENING RADIUS	MODELED DISSIPATION	MEASURED DISSIPATION
$\mathbf{0.5}[\boldsymbol{cm}]$	$0.0031[mol]$	$0.0083 \pm 0.019[mol]$
$\mathbf{1}[\boldsymbol{cm}]$	$0.0033[mol]$	$0.0087 \pm 0.0021[mol]$
$\mathbf{2}[\boldsymbol{cm}]$	$0.0037[mol]$	$0.311 \pm 0.0196[mol]$

To eliminate possible errors due to the heating profile shown in Fig. 2 we applied simpler and better controlled heating profile illustrated in figure 3. Furthermore, to lower the effect from the surrounding of the experimental setup a smaller opening was used for these experiments. The modeled and measured average dissipation of vapor for the second series of experiments is shown in table 2 demonstrating much better agreement compared to the first heating profile, thus, convincing that the deviations between the simulations and experiments are most probably related to uncontrolled air disturbances outside the setup.

Table 2. *Calculated and measured dissipation of vapor for heat profile in the second series of experiments.*

OPENING RADIUS	MODELED DISSIPATION	MEASURED DISSIPATION
$\mathbf{0.25[cm]}$	$0.0021[mol]$	$0.0015\pm0.00067[mol]$

The modeled RH and temperature distributions for the first heating profile at t=251 s. are shown in figure 4 and 5, respectively.

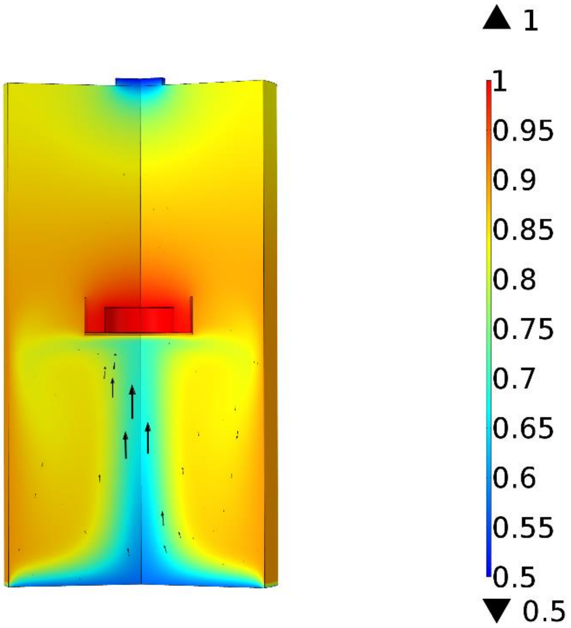

Figure 4: The modeled RH distribution after 251 s for the first heating profile. The black arrows indicate the natural convection.

In figure 6 and 7 the temperature and RH, respectively, for the experiments and simulations are compared This comparison is done for the first heating profile but the trend is observed to be nearly the same for the other measurements and simulations. It is seen that the model underestimates the transient shift in the temperature and therefore also leads to deviation from the experimental data for RH.

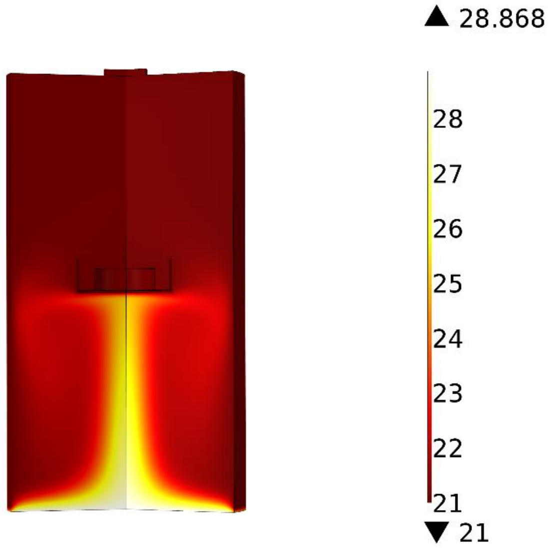

Figure 5: The modeled temperature distribution after 251 s for the first heat profile.

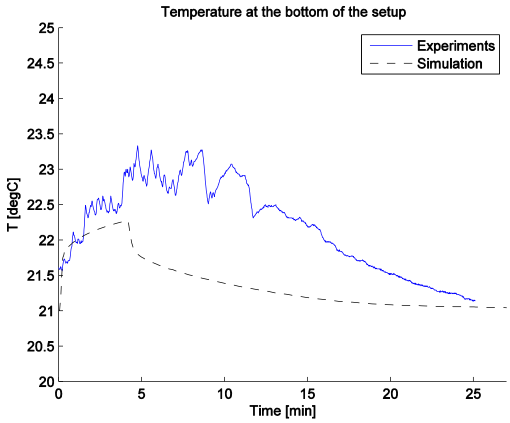

Figure 6: The temperature as a function of time in the bottom of the test setup for the first heating profile.

As mentioned, the model does not account for some convective disturbance from the surrounding of the setup. Thus, the model is expected to, somewhat underestimate the transient behavior of the temperature, since convective surrounding enhances stirring in the setup and rises the heat flux at the bottom. Hence, more energy is released in the setup.

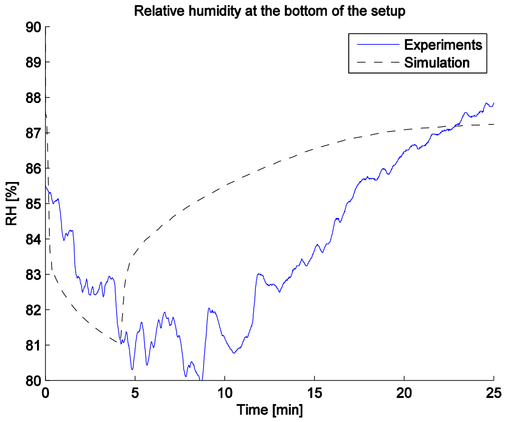

Figure 7: The RH as a function of time in the bottom of the test setup for the first heating profile.

6 Conclusion

The water vapor dissipation was model using the Boussinesq approximation combined the Navier-Stokes equations and the continuity equation for heat and vapor. Experiments provided reasonable agreement with the simulations, at least showing the same order of magnitude values for the vapor flux. The found deviations between the calculated and measured data were related to uncontrolled air disturbances outside the experimental setup.

The presented approach, thus, can, be suggested as a tool to design and optimize packaging technologies with respect to humidity.

Acknowledgements

This work is a part of research activities within the Center of Reliable Power Electronics (CORPE) funded by the Danish Strategic Research Council (currently Innovation Fund Denmark).

References

[1] Ee Hua Wong, Yong Chua Teo, and Thiam Beng Lim. Moisture diffusion and vapour pressure modeling of ic packaging. In *Elec. Comp. amp; Tech. Conf., 1998. 48th IEEE*, pages 1372–1378, May 1998.

[2] EH Wong, R Rajoo, SW Koh, and TB Lim. The mechanics and impact of hygroscopic swelling of polymeric materials in electronic packaging. *Journal of Electronic Packaging*, 124(2):122–126, 2002.

[3] Jiang Zhou and Japhet S Law. Effect of non-uniform moisture distribution on the hygroscopic swelling coefficient. *Comp. and Pack. Techn., IEEE Transactions on*, 31(2):269–276, 2008.

[4] B Xie, XJ Fan, XQ Shi, and H Ding. Direct concentration approach of moisture diffusion and whole-field vapor pressure modeling for reflow process -part i: theory and numerical implementation. *J. Elect.Pack.*, 131(3):031010, 2009.

[5] Haleh Ardebili, Craig Hillman, Marjorie Ann Erickson Natishan, Patrick McCluskey, Michael G Pecht, and Dave Peterson. A comparison of the theory of moisture diffusion in plastic encapsulated microelectronics with moisture sensor chip and weight-gain measurements. *Comp. and Pack. Techn., IEEE Transactions on*, 25(1):132–139, 2002.

[6] H. Bruus. *Theoretical Microfluidics*. Oxford University Press, New York, 2008.

[7] R.B. Bird, W.E. Stewart, and E.N. Lightfoot. *Transport Phenomena*. Wiley International edition. Wiley, 2007.

[8] Jiehei Zhang, Ajaykumar Gupta, and John Baker. Effect of relative humidity on the prediction of natural convection heat transfer coefficients. *Heat transfer engineering*, 28(4):335–342, 2007.

[9] Mark G Lawrence. The relationship between relative humidity and the dewpoint temperature in moist air: a simple conversion and applications. *Bulletin of the American Meteorological Society*, 86(2), 2005.

[10] T.R. Marrero and E. A. Mason. Gaseous diffusion coefficients. *J. Phys. and Chem. Ref. Data*, 1(1):3–118, 1972.