REINFORCING CONCEPTS OF TRANSIENT HEAT CONDUCTION AND CONVECTION
With Simple Experiments and COMSOL Simulations

Sergio Mendez and Lisa AungYong
California State University, Long Beach

Many engineering disciplines require heat transfer in their undergraduate curricula. Students are taught the concepts and mathematics of heat transfer in lecture courses, and this knowledge can be bolstered with hands-on experiential learning activities. It has been reported that more than a third of engineering students learn better with hands-on activities.[1,2] In the literature, simple experimental systems have been developed to enhance teaching of heat transfer. Nassar, *et al*. utilized a flow chamber to calculate the convective heat transfer coefficient, and compared values with known empirical correlations.[3] Clausen, *et al*. reported on free and forced convection for rectangular and spherical objects,[4] and Smart augmented his lectures by heating up rectangular-shaped potato fries.[5] In another study, small modules that fit on a student desk have been developed for students to study transient heat transfer.[6] Some undergraduate chemical engineering laboratory courses offer experiments in heat transfer, but these can be unnecessarily complicated, and often do not have complementary computational modeling. In this paper, we propose a relatively simple module that can be performed as a classroom demo or as a laboratory experiment.

There are many articles in literature that propose computational methods for enhancing the teaching of heat transfer. Partial differential equations encountered in heat transfer problems have been numerically solved using simple tools such as spreadsheets[7,8] or more elaborate programming with Matlab.[9] Hossain, *et al*. measured the temperature profile of a metal fin, and compared the experimental values with numerical solution of partial differential equations and with computer simulations derived from commercially available ANSYS software.[10] The study done in this paper was inspired by the work originally reported by Doughty and O'Halloran where they measured the transient temperature profile from metal and acrylic cylinders, and then performed finite element calculations using spreadsheets and Matlab programs.[11,12] In a previous publication, Mendez, *et al*. presented results on a system similar to the one reported by Doughty and O'Halloran with the added novelty that user-friendly COMSOL Multiphysics was employed to perform three-dimensional (3-D) simulations.[13]

To maintain a competitive edge in today's global economy, chemical engineering students must be taught how to use the

Sergio Mendez *is an assistant professor of chemical engineering at California State University, Long Beach. He received a B.S. degree from U.C. Berkeley and a Ph.D. degree from the University of New Mexico both in chemical engineering. He served four years in the U.S. Marine Corps and was a process engineer at Intel Corporation. His current research interests are primarily in the area of green engineering where he has made significant contributions in research, teaching, and community outreach. His core teaching areas are thermodynamics, separation processes, and the undergraduate laboratories*

Lisa AungYong *obtained B.S. and M.S.E. degrees from the Department of Chemical Engineering at California State University, Long Beach. In her Master's thesis research, she incorporated agricultural "waste" residues into composite foam/hemp sandwich panels. Her goal is to have a successful career as a process engineer in the field of environmental services.*

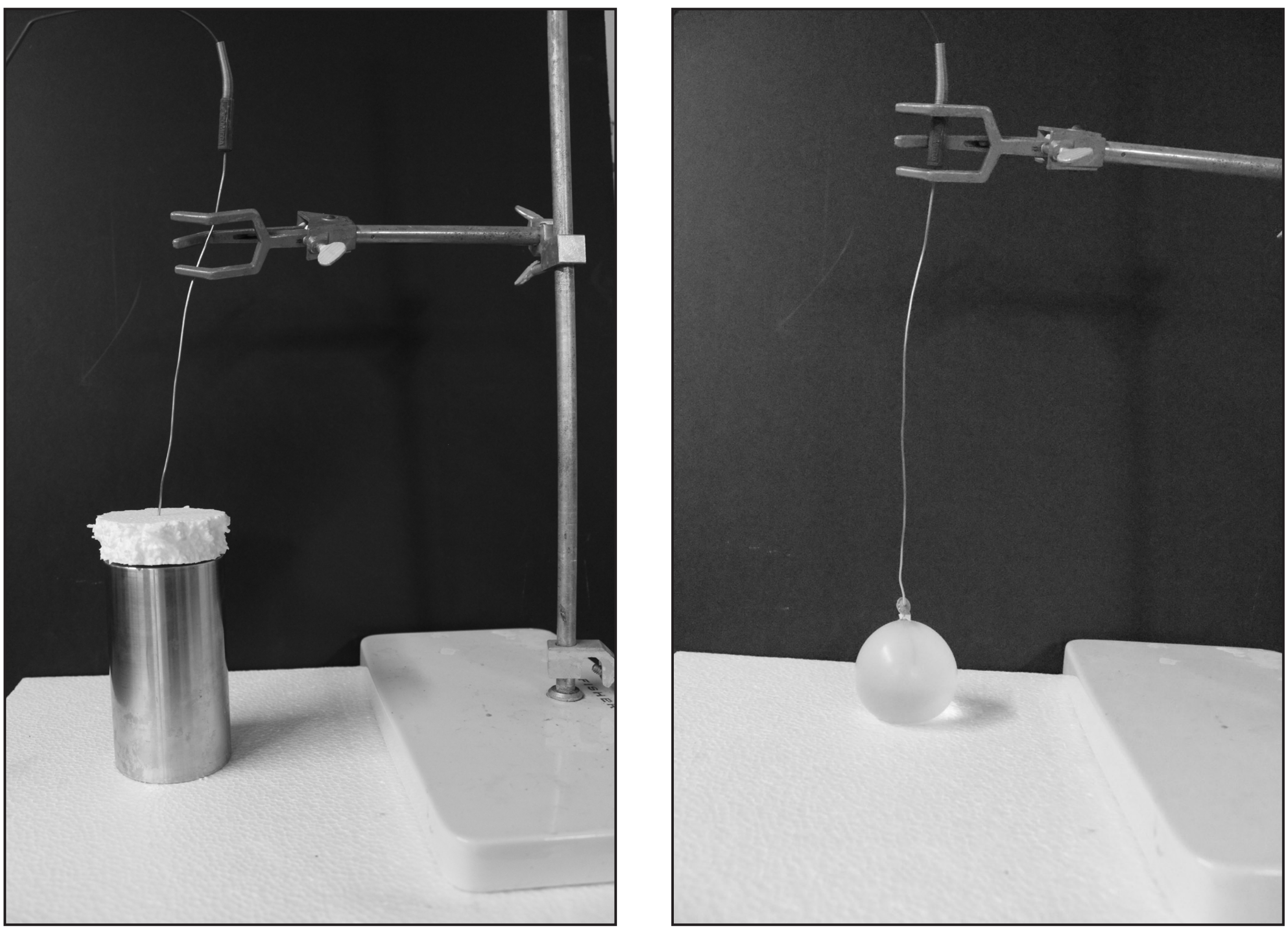
© *Copyright ChE Division of ASEE 2014*

latest technological tools. Since chemical engineering students typically are not trained to write elaborate computer programs, educators should elucidate the capabilities of commercial software that does not require extensive programming with languages such Fortran or C++. The new trend followed by the software vendors is to sell applications that can handle so-called "multi-physics," thereby giving the end user greater modeling capabilities. Since it has capabilities suited for the needs of chemical engineers, we introduce students to COMSOL Multiphysics[14] by having them model transient heat transfer of some simple experimental systems. COMSOL has already been utilized by educators to simulate mass transfer,[15] heat conduction in a fin,[7] and hydrogen fuel cells.[16] Mendez, *et al*. have demonstrated that COMSOL can effectively be used to model free-convection heat transfer from aluminum and plastic cylinders.[13]

In this article, we begin by describing the experimental and computational methods. We then compare the measured temperature versus time profiles of the solid cylinder and the liquid sphere with the profiles from COMSOL modeling. We also present results of student questionnaires that were used to assess the effectiveness of learning transient heat transfer by means of experimentation and computer simulations.

EXPERIMENTAL METHODS

Materials. Objects with cylindrical and spherical geometries were used. An aluminum alloy (7055) solid cylinder was used in the experiments (diameter = 0.0381 m, height = 0.130 m).[11] The density (ρ), heat capacity (C_p), and thermal conductivity (k) of the aluminum are 2710 kg/m^3, 1256 J/(kg · K), and 167 W/(m · K), respectively. For the sphere, water balloons were filled with 0 °C water to a diameter of 0.0365 m for free convection and 0.0476 m for forced convection. The properties of water are ρ =998.2 kg/m^3, C_p=4183 J/(kg · K), and k =0.598 W/(m · K). Figures 1 show photographs of the aluminum cylinder and water balloon. A hole was drilled from the top of the cylinder so that a thermocouple could be inserted to monitor the temperature at half the cylinder height. For the water balloon, the thermocouple was placed in the balloon's opening and tied off using a rubber band.



Figures 1. *Photographs of the aluminum (left) solid cylinder and the balloon filled with water (right). The thermocouples were inserted into the designated holes. The cylinder was placed on a thermally insulated Styrofoam surface, and the top was insulated with a Styrofoam disc.*

Procedure. The cylinder was initially heated to an elevated temperature (~70 °C) in a lab oven. It was then removed from the oven and placed on a thermally insulated foam surface and a foam disk was used to insulate the top surface. A thermocouple was immediately inserted in the drilled hole and the temperature versus time profile was recorded until the solid cooled to ambient room temperature (~24 °C). Since the top and bottom surface were thermally insulated, the heat transferred radially from the cylinder to the cooler room-temperature air via convection. We studied both free convection and forced convection—the latter by blowing air across the cylinder with a fan (air velocity ~ 1 m/s). A similar procedure was employed for the water balloon except that it was initially cooled in a refrigerator to about 0 ˚C, and then set out in the lab to allow its temperature to rise. In this case, heat transferred from the ambient air via convection toward the center of the spherical water balloon.

COMPUTATIONAL METHODS

COMSOL Multiphysics version 4.2a was used to simulate the transfer of heat between the cylinder and sphere and the surrounding air. For the cylinder, we utilized a 3-D half cylinder since we assumed that heat conduction within the cylinder occurred entirely in the radial direction. Although the water balloon used was not completely spherical, we modeled it as a perfect sphere with COMSOL. The dimensions of the objects were drawn to match those of the actual objects. COMSOL employs finite element numerical methods to solve partial differential equations (PDEs). This software conveniently provides PDEs for various phenomena. We used the built-in "Transient Heat Transfer" module in which the software solves the PDE,

$$\rho \cdot C_p \cdot \frac{\Delta T}{\Delta t} = \nabla \cdot (k \cdot \nabla T) + Q \tag{1}$$

where Q is a convective heat flow flux term. This was accounted for with the following equation,

$$Q = h \cdot (T_{ext} - T_s) \tag{2}$$

where T_{ext} is the ambient room temperature, T_s is the temperature at the surface of the object, and h is the convective heat transfer coefficient. We employed the same method as Doughty, *et al*. to approximate the h from experimental measurements[11] where they used

$$\ln\left[\frac{T_c - T_{ext}}{T_i - T_{ext}}\right] = -\frac{t}{\tau} \tag{3}$$

T_c is the transient center temperature (at r = 0), and T_i is the initial temperature. τ is

$$\tau = \frac{\rho V C_p}{h A_s} \tag{4}$$

where V and A_s are the volume and surface area, respectively. The experimental temperature data was plotted as a function of time, t, according to Eq. (3), and a linear fit yielded an estimate of the heat transfer coefficient. To tune the COMSOL model, we adjusted the h value until the simulation results matched the experimental data. Since the COMSOL finite element software solves PDEs, the model required boundary conditions. As noted earlier, for the cylinder, we insulated the top and bottom surfaces with foam panels; therefore, we set such insulating boundary conditions in COMSOL. In addition, since we assumed that heat was transported only in the radial direction, we also applied an insulated boundary condition along the flat surface in the axial direction that split the cylinder in half. This same assumption led to the other boundary condition that there was zero heat flux at the centerline (r = 0). Since the curved surface of the cylinder was exposed to the ambient air, we specified this as the surface where there was convective heat transfer between the warm cylinder and the air. Likewise, for the spherical geometry, we assumed that there was zero heat flux at the center point, and also that the convective heat transfer occurred at the outer surface, which was exposed to ambient air. In addition, the default numerical solver and triangular mesh were used.

RESULTS

In Figure 2 (page 218), we present the experimental and computational results for the cooling of the aluminum cylinder with free and forced convection. For free convection, the temperature dropped slowly from 70 °C to room temperature near 25 °C over a period of about 4.5 hours. As expected, for forced convection, the temperature dropped rapidly from 70 °C to room temperature over a period of about 1.5 hours. These two sets of experimental data were used to estimate the h values that were found to be 10.1 and 47.9 W/(m^2·K) for free and forced convection, respectively. The former value for free convection is in reasonable agreement with the h value reported by Doughty, *et al*.[11] One of the goals of this teaching module was to have students become familiar with the COMSOL Multiphysics software to model this simple experimental system. The results from the 3-D COMSOL simulations are plotted in Figure 2. Within the COMSOL model, we used as a fit parameter the h values, and for the plots shown we found that 10.2 and 38.0 W/(m^2 K) gave the best agreement. For both temperature profiles, there was excellent agreement between experiments and COMSOL indicating that the model is reasonably accurate.

In Figure 3 (page 219), we present the experimental and COMSOL results for warming up of the water balloon with free and forced convection. The experiments were run in a similar manner except the initial temperature of the water balloon was 0 °C. For free convection, the temperature rose slowly from 0 °C to room temperature over a span of 5 hours.

For forced convection, the temperature was elevated faster from 0 °C to room temperature in approximately 3 hours. From this experimental data the h values for free and forced convection were estimated to be 7.4 and 16.6 W/(m^2·K), respectively. For the COMSOL model, we found the best agreement with h values of 11.5 and 15.0 W/(m^2·K) for free and forced convection, respectively. Again, there is reasonable agreement between the experiment and the COMSOL Multiphysics simulations.

COMSOL simulations exhibit good agreement with experimental measurements. Modeling heat transfer with COMSOL Multiphysics has advantages that include user-friendly 3-D modeling, built-in PDE modules for heat transfer, and various tools to plot and visualize simulation results such as colored slice plots and video animations.

ASSESSMENT OF STUDENT LEARNING

In this work we report on a computational model that complements a simple experimental system. The objectives were to 1) enhance students' understanding of free and forced convective heat transfer, and 2) to provide them with an opportunity to learn how to use a modern computational tool. To assess if we met our objectives, students were asked to respond to a questionnaire. The undergraduate students were enrolled in the lecture course "Chemical Engineering (ChE) 420, Heat and Mass Transfer" and they performed this module while in the lab courses "ChE 440, Undergraduate Laboratory I" or "ChE 450, Undergraduate Laboratory II" in the Department of Chemical Engineering at California State University, Long Beach. The total number of students who filled out the questionnaires was 32, which is about 25% of the enrollment. The survey contained eight questions that were measured on a 5-point Likert scale with responses ranging from "Strongly Disagree" to "Strongly Agree" with scores from 1 to 5, respectively. Tabulated are the average scores along with the standard deviations.

As presented in Table 1, the questionnaire results indicate that students felt that lecture only was not sufficient for them to fully grasp the concepts of heat transfer. A large majority of those polled felt that the hands-on measurements were the most beneficial to their learning experience; however, they did have some reservations about conducting COMSOL simulations perhaps because they lacked prior knowledge about the software. Although students expressed a preference

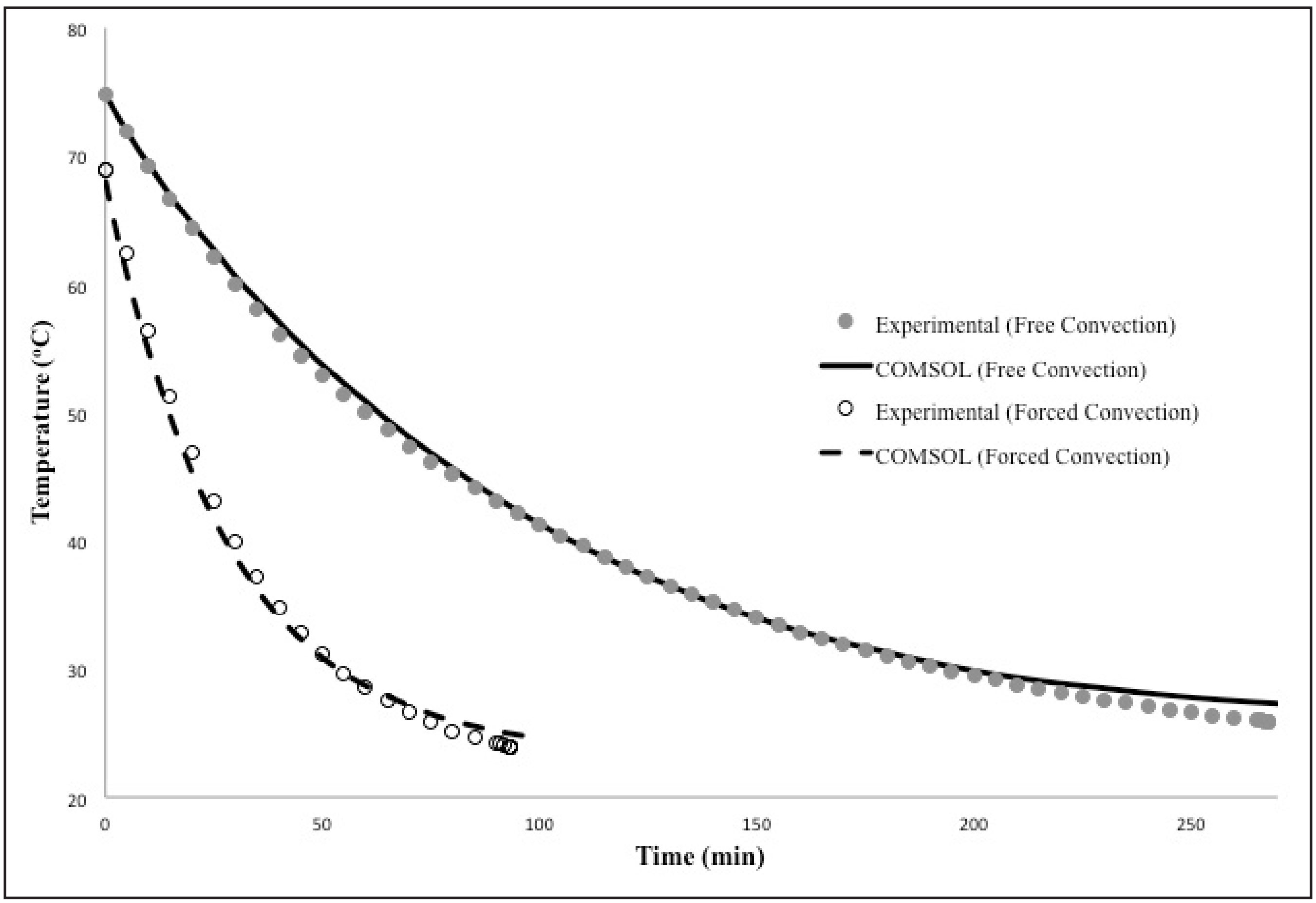


Figure 2. *Temperature profile for aluminum cylinder at centerline radial position. Free and forced convection experimental data as symbols and COMSOL results as curves.*

to not perform the COMSOL simulations, they appreciated the educational value of this module as indicated by responses to questions 5-7. To reduce the added burden of learning how to use the COMSOL Multiphysics software in the future, we plan to introduce the software in lower-division courses, and to provide students with step-by-step custom instructions. We hope that this teaching module with a simple experimental system and complementary simulation can improve student learning about heat transfer with conduction and free and forced convection.

SUMMARY AND CONCLUSIONS

In this paper, we have reported on a teaching module that combines both experimental measurements and computational modeling of heat transfer phenomena. The experimental setup was relatively simple, and the procedure was easy to perform. COMSOL Multiphysics was employed to model heat conduction and convective cooling of an aluminum solid cylinder and liquid water inside

TABLE 1
Results From Student Questionnaire
Responses are averages with standard deviations.

Questionnaire Statement	Response
1. I already understood transient heat transfer based on lecture, textbook and homework, therefore, this module was not necessary.	2.7 ± 0.5
2. Conducting the experimental measurements was the most useful in helping me grasp the concept of transient heat transfer.	4.6 ± 0.5
3. Performing the COMSOL simulations was the most useful in helping me grasp the concept of transient heat transfer.	3.1 ± 1.1
4. Performing this module helped me grasp the concepts of conduction and convection.	3.5 ± 0.5
5. I prefer this type of combined experimental/computational module rather than only experiments.	2.5 ± 1.3
6. Seeing the in-class demo was sufficient for me to better understand transient heat transfer. The lab experiment was unnecessary.	2.9 ± 0.7
7. I would like for the faculty to develop such teaching modules for other chemical engineering courses/labs.	3.7 ± 1.0
8. I plan to learn more about COMSOL Multiphysics and try to perform simulations on my own.	2.4 ± 0.5
1=[strongly disagree], 2=[disagree], 3=[no opinion], 4=[agree], 5=[strongly agree]	

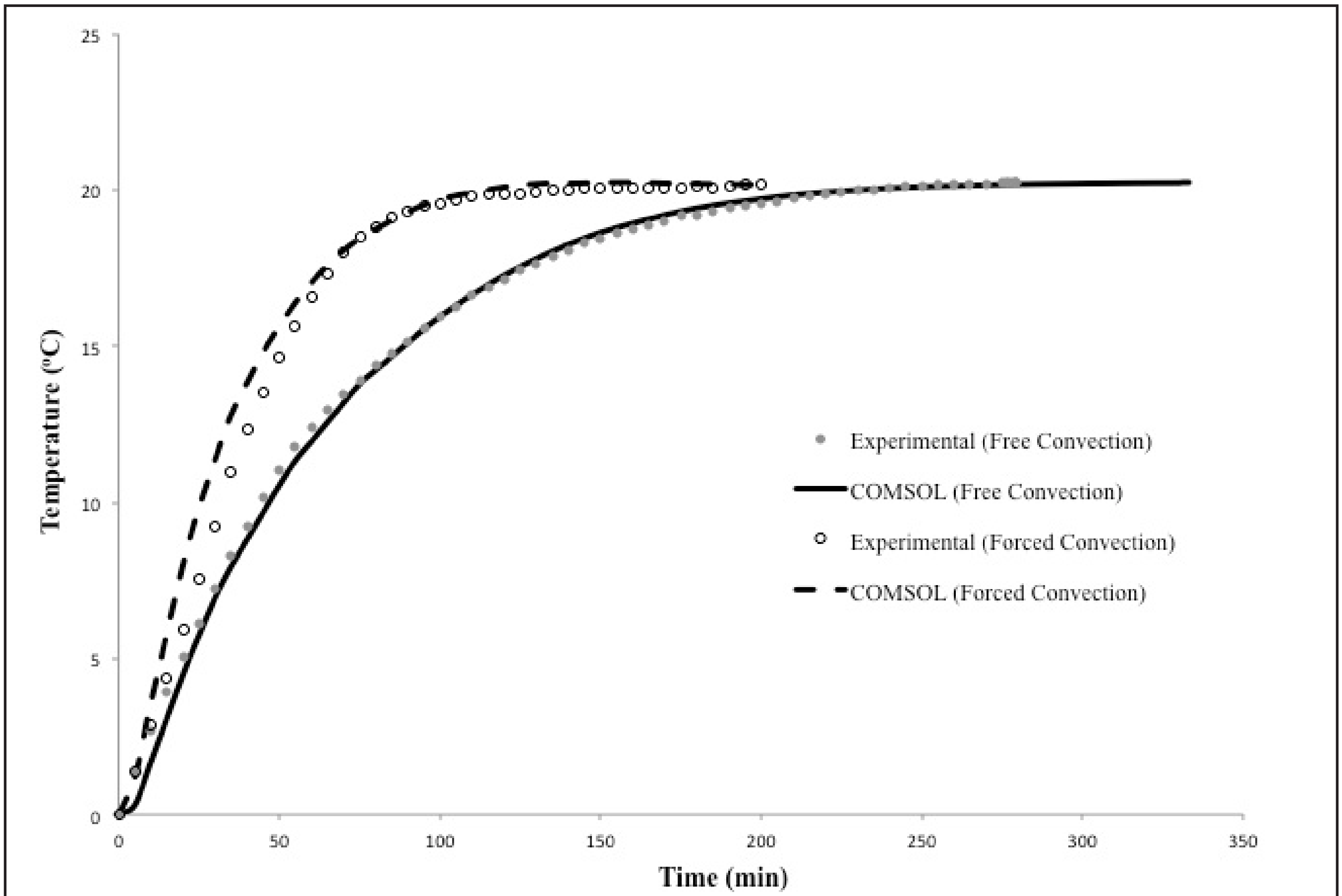


Figure 3. *Temperature profile for the water balloon at the center position. Free and forced convection experimental data as symbols and COMSOL results as curves.*

a spherical-shaped rubber balloon. With the heat transfer coefficient estimated from the experimental data we found reasonable agreement between experiment and computer simulations. From the student questionnaire we found, overall, that students found value in this module, however, learning how to use the computer software require considerable effort. In the future, we hope to introduce students to the software in lower-division courses so that they can become comfortable and proficient in its usage.

ACKNOWLEDGMENTS

We thank Professor Larry Jang and Mr. Minh Tran for helping us with the experimental setup, and Ms. Sophia Nguyen for helping with the initial COMSOL modeling.

REFERENCES

1. Hunkeler, D., and J.E. Sharp, "Assigning Functional Groups: The Influence of Group Size, Academic Record, Practical Experience, and Learning Style," *J. Education*, **86**(4), 321 (1997)
2. Kolb, D.A., *Experiential Learning: Experience As the Source of Learning and Development*, Prentice Hall, Englewood Cliffs, NJ (1984)
3. Nassar, N.N., and A.K. Mehrotra, "Design of a Laboratory Experiment on Heat Transfer in an Agitated Vessel," *Education for Chem. Engineers*, **6**, e83 (2011)
4. Clausen, E., and W. Penny, "Laboratory Demonstrations/Experiments in Free and Forced Convection Heat Transfer Course," American Society of Engineering Education Conference proceedings #1488 (2006)
5. Smart, J.L., "Optimum Cooking of French Fry-Shaped Potatoes," *Chem. Eng. Ed.*, **37**(2), 142 (2003)
6. Minerick, A., "A Desktop Experiment Module: Heat Transfer," American Society of Engineering Education Conference proceedings #1609 (2009)
7. Geike, R., "Comsol Multiphysics in Education-Chemical Reactions, Heat and Mass Transfer," COMSOL Conference proceedings (2008)
8. Besser, R.S., "Spreadsheet Solutions to Two-Dimensional Heat Transfer Problems," *Chem. Eng. Ed.*, **36**(2), 160 (2002)
9. Goldstein, A.S., "A Computational Model for Teaching Free Convection," *Chem. Eng. Ed.*, **38**(4), 272 (2004)
10. Hossain, N.M.A., M.W. Weiser, H.S. Saad, and J.K. Durfee, "Integration of Numerical and Experimental Studies in a Heat Transfer Course to Enhance Students' Concepts," American Society of Engineering Education Annual Conference proceedings #62 (2011)
11. Doughty, T.A., and S.P. O'Halloran, "A Cross Curricular Numerical and Experimental Study in Heat Transfer," American Society of Engineering Education Annual Conference proceedings #1647 (2010)
12. O'Halloran, S.P., and T.A. Doughty, "Integration of Numerical Analysis and Experimental Testing Involving Heat Transfer For a Small Heated Cylinder During Cooling," American Society of Engineering Education Annual Conference proceedings #51 (2009)
13. Mendez, S., and L. AungYong, "Simple Experiments and 3-D COMSOL Simulations to Enhance the Learning of Transient Heat Transfer," American Society of Engineering Education Pacific Southwest Conference proceedings, 501-510 (2013)
14. <http://www.comsol.com>
15. Ortiz, E., J. Vazquez, and L.A. Ricardez, "An Undergraduate Course in Modeling and Simulation of Multiphysics Systems," *Chem. Eng. Ed.*, **44**(4), 299 (2010)
16. Keith, J., F. Morrison, and J. King, "Finite Element Modules For Enhancing Undergraduate Transport Courses: Applications to Fuel Cell Fundamentals," American Society of Engineering Education Annual Conference proceedings #414 (2007) ❒