Energy Research and Development Division
FINAL PROJECT REPORT

SIMULATION MODELS FOR IMPROVED WATER HEATING SYSTEMS

National Lab Buildings Energy Efficiency Research Projects

Prepared for: California Energy Commission
Prepared by: Lawrence Berkeley National Laboratory

OCTOBER 2013
CEC-500-2015-003

BERKELEY LAB
Lawrence Berkeley National Laboratory

PREPARED BY:

Primary Authors:
Jim Lutz
Peter Grant
Margarita Kloss

Environmental Energy Technologies Division
Lawrence Berkeley National Laboratory
1 Cyclotron Road
Berkeley, CA, 94720

Contract Number: 500-10-052

Prepared for:

California Energy Commission

Heather Bird
Contract Manager

Bradley Meister, Ph.D., P.E.
Project Manager

Virginia Lew
Office Manager
Energy Efficiency Research Office

Laurie ten Hope
Deputy Director
ENERGY RESEARCH AND DEVELOPMENT DIVISION

Rob Oglesby
Executive Director

LEGAL NOTICE

DISCLAIMER

The government and the facility operator make no express or implied warranty as to the conditions of the research or any intellectual property, generated information, or product made or developed under this agreement, or the ownership, merchantability, or fitness for a particular purpose of the research or resulting product: that the goods, services, materials, products, processes, information, or data to be furnished hereunder will accomplish intended results or are safe for any purpose including the intended purpose; or that any of the above will not interfere with privately owned rights of others. Neither the government nor the facility operator shall be liable for special, consequential, or incidental damages attributed to such research or resulting product, intellectual property, generated information, or product made or delivered under this agreement

ACKNOWLEDGEMENTS

The project team at Lawrence Berkeley National Laboratory would like to thank the sponsor, the Public Interest Energy Research Program of the California Energy Commission, Heather Bird, and especially Brad Meister.

Thanks to the following individuals who contributed to this work:

Charles Adams, AO Smith

Troy Trant, Rheem

Marc Hoeschele, Davis Energy Group

Carl Hiller, Applied Energy Technology

Moya Melody

Lawrence Berkeley National Laboratory (LBNL) staff: Vi Rapp, Anna Liao, Brett Singer, Robert Cheng, Michael Wetter, Thierry Nouidui, Douglas Sullivan, Peter Biermayer, and Michael Botello.

Thanks, also, to our reviewers for their generosity and time improving this report.

PREFACE

The California Energy Commission Energy Research and Development Division supports public interest energy research and development that will help improve the quality of life in California by bringing environmentally safe, affordable, and reliable energy services and products to the marketplace.

The Energy Research and Development Division conducts public interest research, development, and demonstration (RD&D) projects to benefit California.

The Energy Research and Development Division strives to conduct the most promising public interest energy research by partnering with RD&D entities, including individuals, businesses, utilities, and public or private research institutions.

Energy Research and Development Division funding efforts are focused on the following RD&D program areas:

- Buildings End-Use Energy Efficiency
- Energy Innovations Small Grants
- Energy-Related Environmental Research
- Energy Systems Integration
- Environmentally Preferred Advanced Generation
- Industrial/Agricultural/Water End-Use Energy Efficiency
- Renewable Energy Technologies
- Transportation

Simulation Models for Improved Water Heating Systems is an interim report for the National Lab Buildings Energy Efficiency Research Project (contract number 500-10-052) conducted by Lawrence Berkeley National Laboratory. The information from this project contributes to Energy Research and Development Division's Buildings End-Use Energy Efficiency Program.

For more information about the Energy Research and Development Division, please visit the Energy Commission's website at www.energy.ca.gov/research/ or contact the Energy Commission at 916-327-1551.

ABSTRACT

This research improved the design of computer simulation models of water heaters and hot water distribution systems to increase their accuracy and flexibility, and to allow for future inclusion of new technologies. The models will be incorporated into the Buildings Library at Lawrence Berkeley National Laboratory to assist those performing whole-building energy simulations.

The tasks for the project were to:

- Perform a scoping study and literature review.
- Test water heaters.
- Develop/revise a water heater simulation model.
- Develop/revise a hot water distribution systems simulation model.
- Prepare a report on future potential revisions to California's Title 24.

Laboratory tests were performed on both storage tank and tankless water heaters. Several tests measured heat loss, thermal mass, heat transfer, and mixing. Others focused on gas and electricity consumption and efficiency. Test data were compared against simulation results.

The simulation model for hot water distribution systems estimates energy lost in the distribution system and how occupant behavior affects energy use. A simulation model for generic pipe segments calculates the heat stored within pipes, heat lost to ambient conditions, the temperature of water inside the pipes, and the temperature of water supplied to the end use.

The models developed for this task may find use in supporting revisions to California's Title 24, Part 6, the California Building Energy Efficiency Standards.

Recommendations for future work based on the project include:

- Reviewing the equations and assumptions behind the simulation models.
- Comparing simulation results to recent field-monitoring results.
- Using computer models to confirm calculations.
- Developing code calculations.
- Ultimately, developing compliance software.

Keywords: water heating, water heaters, hot water distribution systems, energy efficiency, simulation modeling, validation, Title 24

Please use the following citation for this report:

Lutz, Jim; Peter Grant; and Margarita Kloss. (Lawrence Berkeley National Laboratory). 2013. ***Simulation Models for Improved Water Heating Systems.*** California Energy Commission. Publication number: CEC-500-2015-003.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

EXECUTIVE SUMMARY

Background

Natural gas fuels nearly 90 percent of California's residential water heating, consuming significant amount of natural gas—nearly 2 billion therms per year. Improved designs for water heaters and hot water distribution systems can help reduce this large energy end use to help California meet its energy and carbon emission-reduction goals in the residential sector.

This research was conducted to improve the design of computer simulation models of water heaters and hot water distribution systems, increase the accuracy and flexibility of the models and allow for future inclusion of new technologies.

Cost savings from improved system designs will provide a robust economic benefit to the state; if the efficiency of all water heaters could be improved to an energy factor of 0.82, the estimated energy savings would be $500 million per year. Improved hot water distribution systems will reduce energy and water use, lower costs, and increase energy efficiency for California residents.

California's energy efficiency requirements for buildings are specified in its *Building Energy Efficiency Standards*, which are developed by the California Energy Commission and regularly updated. Each energy component of a proposed building must meet a minimum stipulated efficiency—including water heaters and hot water distribution systems.

Purpose

This project improved water heating simulation models so they could be included in building energy simulation models, and ensure that they are capable of future expansion to accommodate new technologies.

The simulation tools currently used to model hot water generation and distribution systems are separate, independent models that date from the 1980s to 1990s. Central to this task is to improve and simplify simulation models for modeling water heating equipment and distribution system efficiency. The goal was to ensure that the models could be integrated into Lawrence Berkeley National Laboratory's Buildings Library, where they could be incorporated into whole-building efficiency modeling. The modeling capabilities, once developed, can also inform the Title 24 development process directly or be used to develop specific calculation methods.

Objectives

This project had multiple objectives, each described in a separate chapter.

- Scoping Study: Perform a scoping study to identify current simulation models for water heaters and hot water distribution systems and to recommend strategies for better modeling of those systems.
- Literature Review: Perform a literature review related to simulation models for hot water equipment and distribution systems.

- Laboratory Testing: Perform laboratory tests and use the results to develop simulation models.
- Water Heater Simulation Models: Describe the new simulation models developed for water heaters and the capabilities and limitations.
- Simulation Model for Hot Water Distribution Systems: Describe the new simulation model developed for hot water distribution systems and the capabilities and limitations.
- Future Title 24 Revisions: Describe how the new simulation models could support development of revisions to California's Title 24 building energy efficiency standards.

Conclusions and Recommendations

The conclusions and recommendations are detailed in each report chapter and are specific to the various efforts undertaken. The models and algorithms mentioned are described in detail in the report.

Chapter 1, Scoping Study

The following recommendations grew out of the scoping study and informed subsequent model development aimed at advancing the modeling of water heating and hot water distribution systems.

1. Use techniques developed in earlier simulation models and more recent algorithms, where appropriate, to develop modern, open-source models of water heaters and hot water distribution systems.
2. Use better modeling tools. In particular use the computing language Modelica to develop the new models. Well-documented, easily available source code will speed the adoption, improvement, and capabilities of the models.
3. Create detailed simulation models and compare their results against results generated by earlier models and laboratory testing of selected advanced water heaters.
4. Consider using the Building Controls Virtual Test Bed software or the Functional Mock-Up Interface to connect the water heating and hot water distribution system simulation models to models for simulating residential building energy. This connection will enable simulations to capture interactions among systems that are now treated independently. For example, how does the warm, moist air created by a resident's shower affect a home's heating, ventilation, and air conditioning (HVAC) system?

Chapter 2, Literature Review

This literature review identified studies and reports that contain algorithms and equations to model components of water heater and hot water distribution systems. These algorithms and equations serve as the foundation for developing the simulation models later in this task.

The algorithms described in the Arthur D. Little model serve as the starting point for modeling storage water heaters. The algorithms used by Grant and by Burch et al. are the foundation of the simulation models for tankless water heaters. Improvements to the original models will be based on techniques described in the reviewed literature.

For the simulation models of hot water distribution systems, the key algorithms are based on the models implemented in HWSIM (Hot Water Simulation, a detailed event-based simulation model) and the Oak Ridge National Laboratory model. The algorithms are modified to account for the dynamics of the delivery phase of draw events, as detailed by Hiller's studies.

Chapter 3, Laboratory Testing

This chapter describes the laboratory tests performed to develop simulation models and the test results. In some cases the data could be enriched to provide more useful information. The following are examples of how this testing effort could be continued.

- Only one condensing storage tank water heater and one non-condensing tankless water heater were tested. Those numbers do not enable researchers to assess differences among manufacturers. Performing the same tests on additional heaters from different manufacturers would expand and enrich the available information.
- No condensing tankless water heaters or hybrid water heaters were tested. Condensing tankless and hybrid water heaters, which are considered up-and-coming technologies, may become a significant portion of the market. Future efforts should focus on testing those types of heaters before their popularity increases.

Chapter 4, Water Heater Simulation Models

Lawrence Berkeley National Laboratory's water heating simulation models enable researchers to study questions that have previously been unanswerable. Some of the studies that are now possible are briefly described briefly.

- Because the water heating section of the current Title 24 is not based on simulation studies, there is no way to identify its effectiveness. Simulation studies, using Lawrence Berkeley National Laboratory's new models, can support significant increases in water heating energy efficiency throughout California.
- Currently the in situ efficiency of tankless water heaters is a major discussion topic in the hot water community. Tankless water heaters are not as efficient as predicted by the U.S. Department of Energy's energy factor test; however, no significant study has been performed to identify the effect of draw profile on efficiency. Performing such a study would enable researchers and code developers to better identify an effective efficiency derate factor and allow installers to better determine whether to install a storage tank or tankless water heater.
- Base models are now available to study the best way to redesign water heaters. By changing parameters or quickly modifying control logic, simulation can identify how much energy could be saved by adding insulation to water heaters, decreasing

capacitance of tankless water heaters, or varying the control strategy. Such studies can inform manufacturing decisions or be used to require higher-efficiency models, and thus drive design changes.

Chapter 5, Simulation Model for Hot Water Distribution Systems

Simulation models for studying a hot water distribution system are tools the hot water research community requires to continue to improve the understanding of residential hot water systems. New models enable users to simulate the piping distribution system, fixtures in a building, and occupant behavior, which will enable researchers to investigate several significant topics.

1. By predicting the delay time between the start of a draw and hot water reaching the fixture, researchers can predict occupant satisfaction with various plumbing configurations.

2. As water passes through the distribution pipes it loses heat to the environment, causing a decrease in the temperature of the water that reaches the fixture. In some cases, the losses may be significant enough that the water reaching the fixture is not hot enough to satisfy the occupant. Researchers can now perform simulations to identify when such situations may occur.

3. One major current research topic concerns how much hot water remaining in the pipes is still useful at the start of the next draw. Because the model simulates heat losses to ambient conditions and the temperature of the water in the pipes between draws, researchers will be able to identify the temperature of water in the pipes at the start of a draw. This knowledge makes it possible to characterize the amount of heat in the pipes that is still useful at the start of any given draw.

4. EndUseTwoBranch models (described in the report) enable researchers to include user actions in models. Researchers can model occupant behavior by stating the flow rate and temperature the user wants at the outlet of the fixture. This capability allows modelers to simulate the impact of consumer behavior on water and energy waste.

5. Because the hot water distribution system model was created in the Modelica environment (as were the water heater models), all the models necessary to study a house's complete hot water system are available in the same location. Those models can be used to quickly construct a hot water distribution system by dragging and dropping the appropriate components. Because the Lawrence Berkeley National Laboratory Buildings Library is also available in Modelica, it will be possible to include hot water systems in whole-building simulations.

Chapter 6, Future Title 24 Revisions

Developing the 2016 Title 24 standards already has begun.

- Now is a good time to systematically review the equations and assumptions behind the hourly adjusted recovery load. The robustness of the factors and terms in those calculations should be subjected to uncertainty analysis.

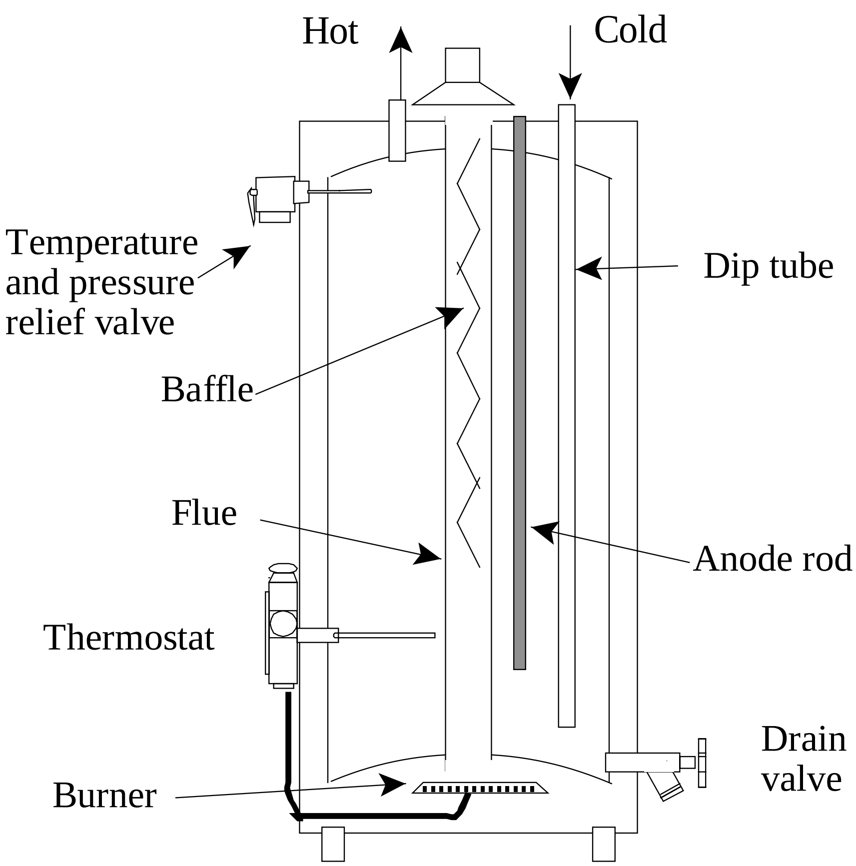
- Many of the Energy Commission's research and development projects could offer approaches and results for improving the calculations for water heating energy use. The effect of different sizes and layouts of hot water distribution systems could be investigated further using the new computer models.
- The hourly energy consumption predicted from the water heating energy use calculations in Title 24 could be retroactively compared to results of recent field monitoring.
- The simulation models for water heaters and hot water distribution systems still contain significant shortcomings. Longer-term efforts to improve the models should address computational speed; validation of results; and quality (open, reliable, efficient, and maintainable source code). The ability to quickly model different types of advanced water heaters is important, as is the ability to model recirculation systems and drain water heat recovery.
- Improve the simulation models so they could be used as compliance software for Title 24 requirements.

CHAPTER 1: Scoping Study

Researchers conducted a scoping study of water heater and hot water distribution system energy simulation models. Simulation modeling strategies were considered.

The typical water heater in California is a gas-fired, natural draft storage water heater. Figure 1 shows a schematic of this type of water heater.

Figure 1: Typical Gas-Fired Storage Water Heater



1.1 Current Water Heater Simulation Models

There are several computer simulation models for water heaters. The most comprehensive general models are TANK and WATSIM. Other water heater models have been built using TRNSYS and similar general-purpose computer simulation tools. Although solar water heating is not the subject of this task, extensive modeling of water heaters has been done in connection with studies of solar water heaters. Another area with extensive modeling work are heat pump water heaters (Guo et al. 2011). Also of interest is widespread research on heat pump water heaters currently being conducted at universities in China. Extensive literature exists on some aspects of modeling water heaters, such as stratification in storage tanks (Hiller et al. 1994; Cragen 1994). Specific modeling topics relevant to this project are described in more detail in Chapter 2, Literature Review.

Several water heating simulation models are described below.

1.1.1 TANK

TANK was developed in the early 1990s to perform detailed thermal analyses of storage-type water heaters (Paul et al. 1993). The software models a typical center flue, gas-fired, storage-type water heater. The model was developed by the Gas Research Institute (GRI) in response to a U.S. Department of Energy (U.S. DOE) rulemaking to revise the minimum efficiency performance standard for residential water heaters (U.S. DOE 2001). TANK numerically predicts the performance of water heaters during the U.S. DOE 24-hour simulated use test for water heaters (U.S. DOE 1998). TANK uses a finite-difference algorithm having variable-length time steps to calculate temperature and energy flows among 20 nodes in the water in and materials of the tank.

TANK was written in BASIC for DOS operating systems before the development of Windows. The code is available, although it was not well documented. Deciphering the algorithms used in the model is difficult. Incorporating the code into modern software also has been problematic (Lutz 2010).

TANK is designed to model only central-flue storage-type water heaters. It is unable to model condensing, tankless, or other recent innovative gas-fired water heater designs, or electric water heaters of any type.

1.1.2 WATSIM

WATSIM is another general-purpose water heater simulation model. It was written for the Electric Power Research Institute (EPRI), also in the early 1990s (Hiller 1994). In addition to being a simulation model, it has the capability to create residential water draw profiles given customer demographic information. WATSIM is applicable to vertical-cylinder, tank-type water heaters. The tank is modeled quasi-three-dimensionally using two-dimensional calculations with radial symmetry. The tank can be divided into as many as 24 vertical zones with 12 heat inputs. WATSIM also uses dynamically changing time steps during the simulation. The mixing of hot and cold water in the tank during draws and during reheat are modeled using empirical correlations. Those empirical correlations, as well as the code itself, are proprietary and not available for examination.

Although not relevant to this scoping study, concerns about the ability of the water draw generator have been raised. Apparently the timing for the draws is not truly random (Cragen 1994).

1.1.3 HEATER

Another earlier water heater computer model was developed by Arthur D. Little for U.S. DOE in 1982 (ADL 1982a). The program was written in FORTRAN, and the input files were punched computer cards. By default, the model divides the tank into six vertical sections. An energy and mass balance is performed on every section. The calculations include the effects of buoyancy-induced water currents inside the tank. A variable time step also is used in the model. Although the code for this model is not available, the algorithms used are well documented in the report.

1.1.4 TRNSYS

Many other water heater simulation models have been developed for specific studies but have not achieved widespread use. See, for example, Ellul and Muscat 2008. Many of those models use the TRNSYS program developed by the Solar Energy Lab at the University of Wisconsin. As described on the TRNSYS website,

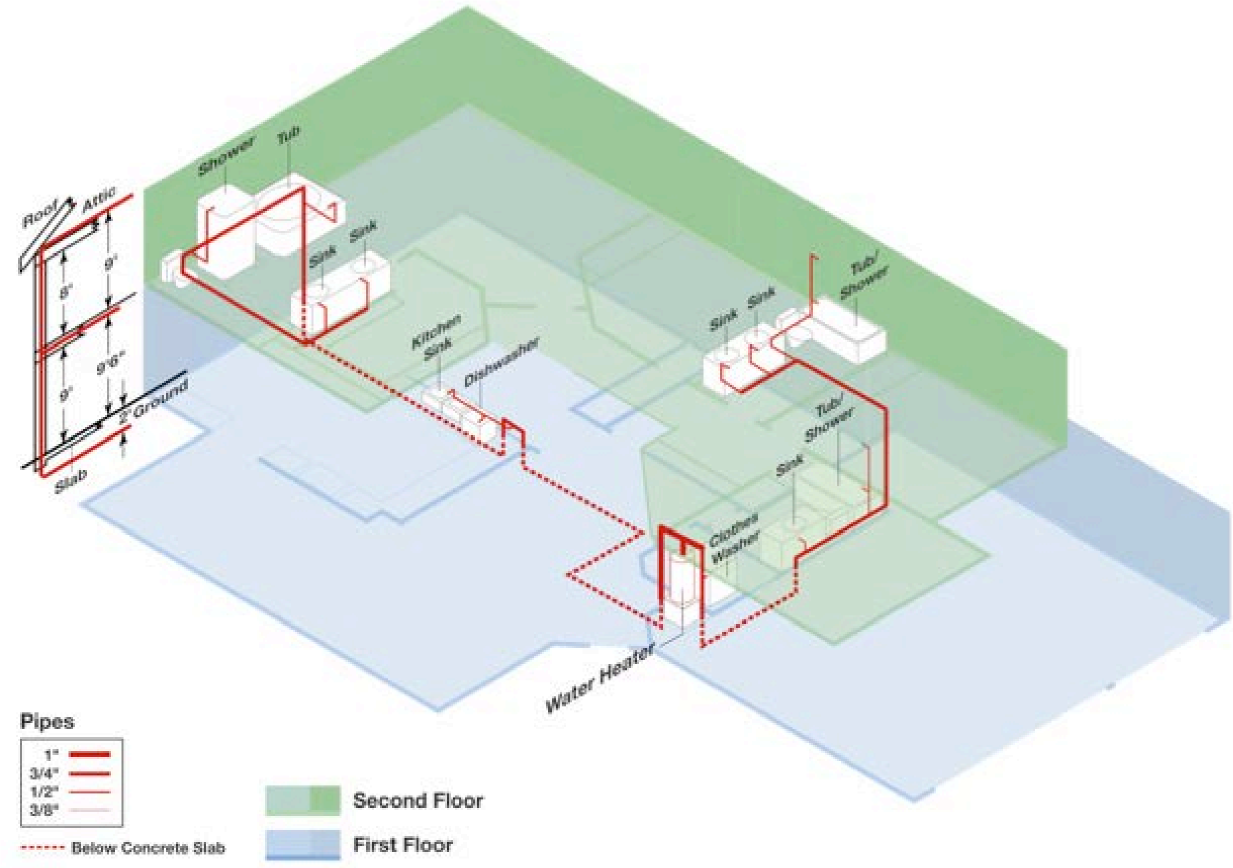
> TRNSYS is a transient systems simulation program with a modular structure. It recognizes a system description language in which the user specifies the components that constitute the system and the manner in which they are connected. The TRNSYS library includes many of the components commonly found in thermal and electrical energy systems, as well as component routines to handle input of weather data or other time-dependent forcing functions and output of simulation results. The modular nature of TRNSYS gives the program tremendous flexibility, and facilitates the addition to the program of mathematical models not included in the standard TRNSYS library. (University of Wisconsin 2011)

One primary application has been solar thermal systems. TRNSYS includes components for modeling storage tanks of various geometries and heating sources (Thermal Energy System Specialists 2010). The core code of TRNSYS is written in FORTRAN (Bradley and Kummert 2005).

1.2 Current Models of Hot Water Distribution Systems

Figure 2 shows a traditional California residential hot water distribution system. This configuration of a trunk-and-branch water plumbing system shows two primary trunks serving different parts of the home. Note that parts of both trunks are below slab, as indicated by a red dashed line; pipes below slab usually were not insulated. A recent alternative to this installation includes home-run or manifold systems, where a hot water line is run directly to each end use. Also becoming common is a mixed or mini-manifold system that has a local, smaller manifold serving each cluster of end uses such as a bathroom or kitchen. Occasionally recirculation loops are used to pump hot water in a loop to each end use and then back to the water heater.

Figure 2: A Typical Hot Water Distribution System



Several models of hot water distribution systems (HWDS) have been developed. A few of the models are described below.

1.2.1 HWSIM

The HWSIM water heating system model was developed by Davis Energy Group for the California Energy Commission (Davis Energy Group 1990). The model enabled distribution systems to be modeled either as a trunk-and-branch system or as a home-run system. The temperature decay is calculated in each section of the distribution system during the times between draws. In 2004 Davis Energy Group received additional funding from the U.S. DOE's Building America program to develop an enhanced version of HWSIM with improved capabilities and a graphical user interface. HWSIM was updated for the PIER Program (Davis Energy Group and Scott 2007). Improvements made to the model enable users to change environmental temperatures (inlet water and ambient) and apply enhanced heat loss algorithms. The user interface also was improved. Further work was performed using HWSIM for the U.S. DOE's Building America program, including adding the capability to model recirculation systems.

Recently TANK was combined successfully with HWSIM (Lutz 2010). Because TANK's computer code is relatively old and poorly documented, it has been difficult to merge the code into modern software environments. In the current version, the TANK part of the combined program is very slow. The 2011 Building America funding also allowed for integrating pipe radiant heat transfer algorithms. This model was then validated against laboratory heat loss data and found to be in good agreement (Hoeschele and Weitzel 2012).

1.2.2 ORNL Hot Water Distribution System Model

Oak Ridge National Laboratory (ORNL) developed a simulation model for studying hot water distribution systems. The model was written in LabVIEW, a graphical programming language most often used for data acquisition and analysis in laboratory settings. The model estimated the heat loss or gain from insulated and uninsulated hot water pipes. The model was used to evaluate the relative consumer cost of a range of plumbing design options and alternatives for the Energy Commission (Wendt et al. 2004). Costs were derived from an analysis of expected utility cost savings. The study focused on two types of draw patterns: one involving clustered draws and the other involving draws from a cold start. These cases are extremes—best and worst cases—of actual hot water use patterns, and provide limits to the analysis.

1.2.3 National Association of Home Builders (NAHB)

The National Association of Home Builders Research Center (NAHB) developed an HWDS simulation model to help evaluate the impact of various plumbing configurations on the performance of tankless water heaters (Wiehagen and Sikora 2002). The simulation model is described only briefly in their report.

1.2.4 Florida Solar Energy Center (FSEC) - HWDS

A simplified hot water distribution system model suitable for inclusion in a DOE-2 building energy simulation model was developed at the Florida Solar Energy Center (FSEC) (Gu 2007). This simple model, however, is not suitable for modeling branching systems having more than one end-use point.

1.2.5 American Society of Heating and Air-Conditioning Engineers (ASHRAE)

An earlier study of the distribution efficiency of startup and shut-down flow in hot water lines, while technically not a simulation model of hot water distribution systems, must be mentioned as a good source of modeling algorithms (Schultz and Goldschmidt 1983).

1.3 Modeling Strategies

By scoping and deconstructing current models, the project team was able to compare equations, better understand modeling techniques, and determine various model strengths and shortcomings. The goal was that the water heater and hot water distribution system models developed in this task offer the following benefits:

- Be capable of being integrated into the building simulation models.
- Use open source programs to encourage future expansion and continuous improvement.
- Use modular code, enabling others to easily add new calculations and features.
- Allow for easy expansion and upgrades to accommodate future technologies.

To meet the above goals, project team chose to use the Modelica modeling environment.[1] Modelica is a non-proprietary, object-oriented, equation-based language. With the goal of convenience, it uses differential, algebraic, and discrete equations to model the dynamic behavior of complex physical systems consisting of components, such as mechanical, electrical, electronic, hydraulic, thermal, control, and electric power. Model developers can use Modelica to build models of water heaters and hot water distribution systems out of models of simpler components or individual features.

The models developed in this project are linkable to building energy simulation models with either the Building Controls Virtual Test Bed[2] or the Functional Mock-Up Interface.[3]

---

[1] Modelica website: https://modelica.org/

[2] Building Controls Virtual Test Bed website: http://simulationresearch.lbl.gov/bcvtb/

[3] Functional Mock-up Interface website: http://www.functional-mockup-interface.org/

CHAPTER 2: Literature Review

The literature review identified the optimal algorithms and equations to use to model components of water heater and hot water distribution systems. After briefly discussing the literature on models of water heater and residential hot water distribution systems, this chapter summarizes the literature review process.

2.1 Water Heater Models

Water heater models, as described in Chapter 1, include:

- TANK, developed by the Gas Research Institute
- WATSIM, developed by the Electric Power Research Institute
- TRNSYS, developed by the Solar Energy Lab at the University of Wisconsin
- HEATER, developed by Arthur D. Little for U.S. DOE

The three most detailed general-purpose water heater simulation models used for calculating energy use are TANK, WATSIM, and HEATER.

2.1.1 TANK

As described in Chapter 1, TANK was developed to perform detailed thermal analyses of typical center-flue, gas-fired storage-type water heaters only (Paul et al. 1993). TANK numerically predicts the performance of water heaters during a 24-hour simulated use test. It uses a finite difference algorithm having variable-length time steps to calculate the temperature and energy flows among 20 nodes in the water and materials of the tank. The code for TANK is available, but it is difficult to decipher the algorithms, which were not documented. Reverse-engineering a full water heater model from TANK could be costly.

2.1.2 WATSIM

WATSIM is a general-purpose water heater simulation model written for vertical-cylinder, tank-type electric resistance and heat pump water heaters only (Hiller 1994). The tank is modeled quasi-three-dimensionally using two-dimensional calculations with radial symmetry. Mixing hot and cold water in the tank during draws and reheat is modeled using empirical correlations. Those empirical correlations, as well as the code itself, are proprietary and not available.

2.1.3 HEATER

Another early water heater computer model was developed for the U.S. DOE (ADL 1982b). By default, the HEATER model divides the tank into six vertical sections. An energy and mass balance is performed on each section. The model assumes buoyancy-driven water flows and energy transfer off the water side of the heated bottom of the tank and off the wall of the flue. Additionally, a "recirculation flow," or stirring action, is assumed in the zones where there is active buoyancy flow (the cold water below the thermocline). The buoyancy flow rises until it hits a so-called stagnant region above a thermocline, where the buoyancy flows stop. The equations and algorithms in the report are clearly, if succinctly explained. Figure 3 shows a general schematic of the model used.

Figure 3: The HEATER Model

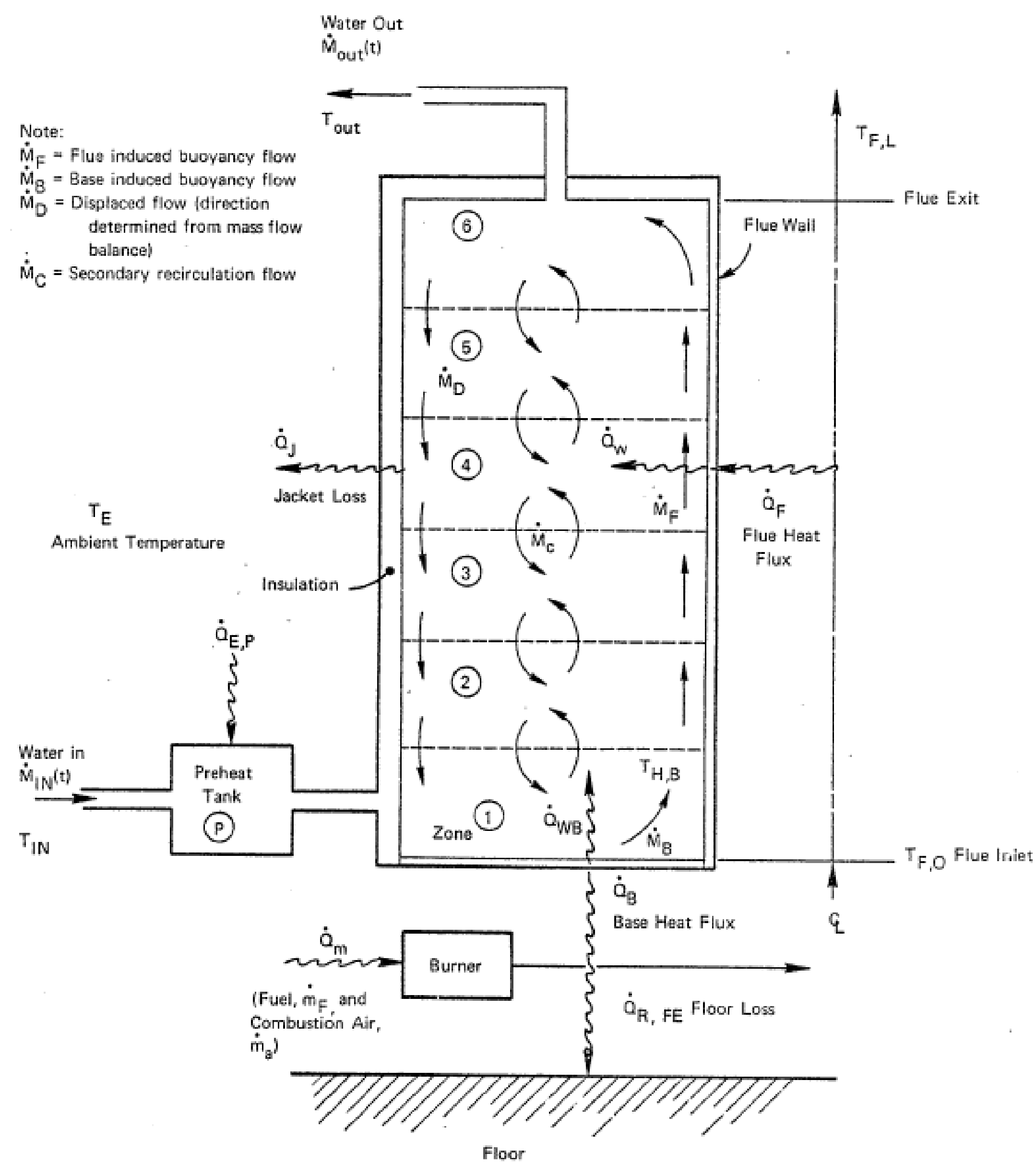
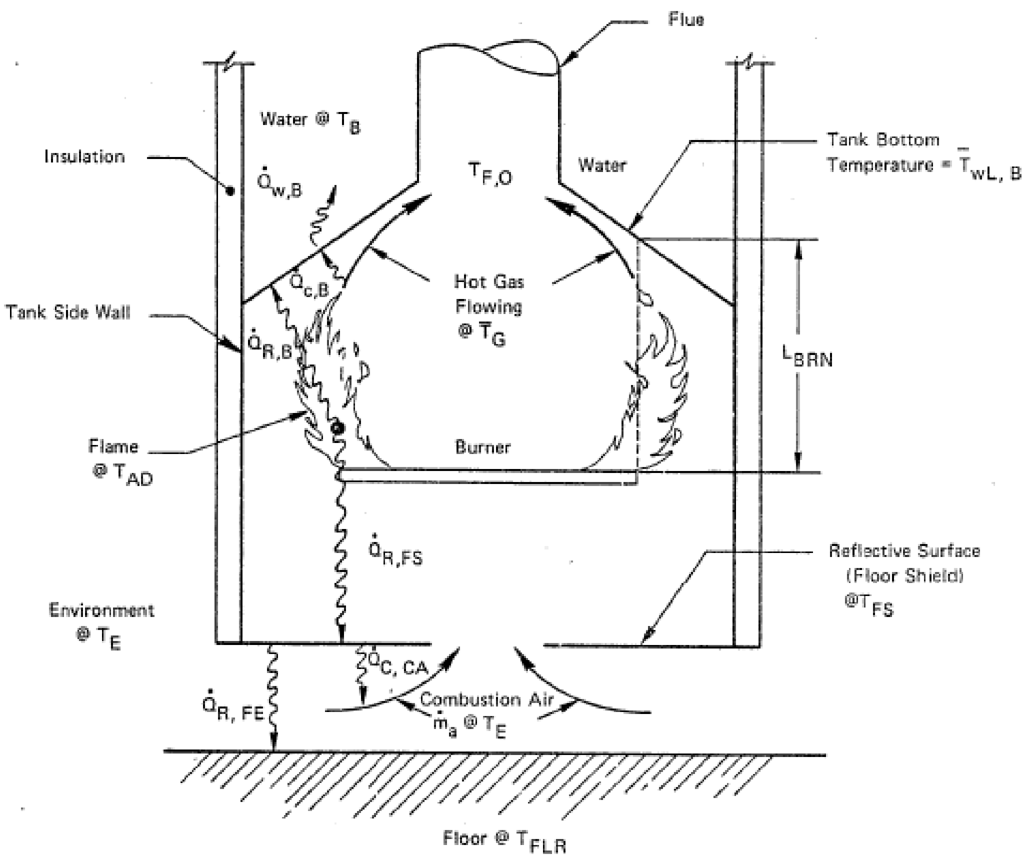


Figure 4, a similar schematic drawing, shows the details of heat flows and temperatures of the burner flame, the combustion chamber, and the bottom of the flue and tank.

Figure 4: Details of Heat Flows and Temperatures in HEATER



Source: A.D. Little

Some of the heat transfer coefficients used in HEATER, however, appear to be based on research that was not applied appropriately. The heat transfer coefficient used for heat transfer from the base, for example, appears to have been taken from research on heat transfer from heated flat walls, not a flat surface (MacGregor and Emery 1969).

2.1.4 Other Relevant Research

Goldschmidt undertook a series of investigations of the effect of baffles on improving the heat transfer from combustion gases in a vertical flue in gas-fired water heaters (Goldschmidt and Beckermann 1986; Goldschmidt and Offhaus 1987; Goldschmidt et al. 1989).

The heat transfer coefficient from flue wall to the water in the tank can be improved by using the results of experiments on natural-convection heat transfer from the outer surface of a vertical cylinder to liquids by Fujii et al. 1970.

The work of Consul et al. 2004, showed how the advent of computational fluid dynamics could add possibilities for better understanding the heat and mass transfer through virtual prototyping of water heaters. The work of Hmouda et al. (2010) represents a recent example of this type of analysis as applied to natural convection cooling of a center-flue gas-fired storage water heater. The results obtained from such studies could be used to develop correlations to feed simplified models.

The mixing of water during draws affects the stratification of the water in the tank. This effect is important in determining how well the hot water is kept unmixed with the incoming cold water. Zurigat et al. (1988) have performed studies of thermocline development in storage tanks using salt concentration to emulate the density differences due to heating.

Plume entrainment is another mechanism that can compromise stratification in storage tanks. A plume of hot water rising through a region of cold water causes some mixing, both of the plume and water it rises through. Kleinbach et al. (1993) found this to be an important issue for solar water heating.

Condensing gas-fired storage water heaters use helical coiled flues. The heat convection coefficients on the water side of those components can be derived from the experimental investigation of natural convection from vertical helical coiled tubes by Mohamed (1994).

Research on heat transfer from condensing flue gases usually originates from studies of electrical power plants, where the flue gas is passed outside of tubes that have fluid inside. Condensing storage water heaters operate in reverse. Condensation of combustion products occurs on the inside of coils that are cooled by the water in the tank. An extensive study of the heat and mass transfer from condensation of water and acids from combustion products of natural gas is found in Valencia et al. (2004).

With tankless water heaters, the water is not stored but rather heated directly as it is used, typically in a fin-tube heat exchanger over a burner. A simple model of a tankless water heater was developed by Burch et al. (2008) and Grant et al. (2010) using a lumped node for the combined mass of the heat exchanger and water, with coupling to environment, draw loss, and gas input.

Modeling for a condensing tankless water heater must consider that the combustion gases do not change temperature as the latent moisture in the combustion products condenses. The heat transfer occurs at a constant temperature as the moisture changes state from a vapor to a liquid. Techniques to model this transfer of heat and mass in condensing instantaneous boilers are described by Idem et al. (1992).

2.2 Models of Hot Water Distribution Systems

Heat transfer during steady-state operation—for example when hot water is flowing through a pipe—is well covered by standard engineering calculations. Modeling the temperature decay of non-flowing hot water in a pipe after a hot water draw has ended is extensively discussed by Schultz and Goldschmidt (1983). They also provide careful treatment of calculating the step-change in temperature under steady flow conditions. It is not clear, however, that this is an adequate model of the more common step-change from no-flow cool water to flowing hot water conditions.

Hot water use in residential buildings is a dynamic and stochastic process. Hot water draw events comprise three stages: (1) delivery, as cool water in the pipe is pushed out by the incoming hot water; (2) use, when steady temperature and flow have been reached as the water is used for its intended purpose; and (3) cool-down, when the hot water remaining in the pipe after the draw cools down (Klein 2006). In some cases such as shower, because of user behavior, there may even be a fourth stage, adjustment that occurs after initiation and before use. This is when a user is adjusting the flow of hot and cold water so the combined flow meets the desired temperature and flow rate (Lutz 2012). A simulation model of segments of pipe in an HWDS must be able to handle all these cases. In a branching HWDS containing multiple end-uses, multiple simultaneous draws occasionally occur. In those cases, the stages of use of a hot water draw for an individual may be interrupted by another draw.

Simulation models of hot water distribution systems are discussed in Chapter 1. The HWSIM model is reviewed here.

2.2.1 HWSIM

The HWSIM water heating system model was developed for the Energy Commission as a detailed water heating method to support improvements to California's Title 24 (Davis Energy Group 1990). The hot water distribution piping layout to each use point is modeled as a tree, with the water heater serving as the base of the tree.

The simulation calculations are based on heat loss from small volumes of water as they travel through the HWDS. The heat transfer modeling is based on engineering principles. The temperature decay is calculated in the main and branch sections of the distribution system during the periods between draws.

The HWSIM model has been updated and improved over time (Davis Energy Group and Scott 2007; Springer et al. 2008; Hoeschele and Weitzel 2012). Improvements enable changing environmental temperatures (inlet water and ambient), enhanced heat loss algorithms, and a graphical user interface. The capability to model recirculation systems has not yet been implemented.

Recently TANK was combined successfully with HWSIM (Lutz 2010). The combined software has been used for analyses of residential hot water distribution systems for the Energy Commission's revision of Title 24. Because TANK's computer code is relatively old, the TANK part of the combined program is slow.

2.2.2 Other Relevant Research

Work at the University of Victoria recently has developed a hot water system model based on MatLab and Simulink (Li 2011; Schneyer 2011). The algorithms used to model the components of the HWDS are not documented in detail, however.

Gu (2007) provides a detailed discussion of algorithms on a simplified hot water distribution system model. That report does not consider branching pipes, however, nor the mixing that occurs as hot water pushes cooled-off, previously hot water out of the pipe at the beginning of a draw. This mixing occurs many times a day and has a significant effect on user comfort and water use.

Baskin et al. (2007), of ORNL, provide a detailed discussion of the algorithms and analytical procedures for calculating heat flows and temperatures along a series of pipe segments. The water inside the pipe segments is modeled one-dimensionally in the axial direction. The heat flow out of the pipe segments is modeled radially. The source code of the core calculation program was written in C++.

The flow of hot water into a pipe filled with still, cooler water is a complex, dynamic process. In nearly all cases the interface between the cold and hot water is not a flat plug. And under a surprising number of conditions it is not the parabolic shape expected by turbulent flow as considered in standard fluid dynamics theory. Those dynamic conditions have some unexpected results on heat loss and time for hot water to arrive at the end use. Hiller (2006a and b; 2008a and b; 2011a and b) has experimentally investigated the heat loss factors and delivery times of hot water when a hot water draw starts from no-flow conditions.

Chapter 3
Laboratory Testing

Lawrence Berkeley National Laboratory (LBNL) performed experiments on several different types of water heaters. As described in chapters 1 and 2, data previously collected for older models of water heaters were unable to answer research questions that have arisen since the data were collected. This chapter presents the results of experimental efforts to collect the background data necessary to create the next generation of water heater simulation models.

The goal of this work was to improve the base of experimental data for use in developing simulation models for several kinds of water heaters, both common and advanced. Tests were performed on, and data are available for, three non-condensing storage tank water heaters, one condensing storage tank water heater, and one non-condensing tankless water heater. The measured data enable researchers to study how the heaters behave, create and validate simulation models, and undertake deeper analysis of the efficiency of the units.

3.1 Storage Tank Water Heaters

Non-condensing storage tank water heaters are the most prevalent form of gas-fired water heater on the market today. They tend to consist simply of a storage tank filled with water, a combustion chamber beneath the water, and a central flue. The controls are a simple thermostat with a set temperature and deadband installed at a certain height in the water volume. The set temperature is user controlled, whereas both the deadband and height of the thermostat are determined by the manufacturer.

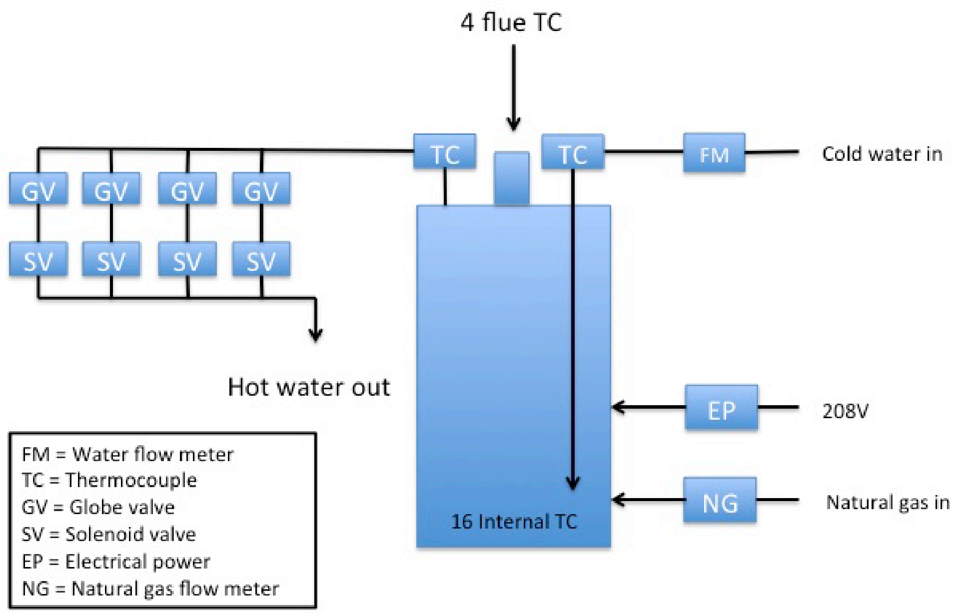
The goals of testing storage tank water heaters are listed below.

- Generate the data necessary to support the understood processes in new models. These data pertain to burner operation, pilot light operation, and thermostat operation.
- Collect data needed to study the some processes that are either poorly understood or differ from heater to heater. Those processes include mixing in the tank caused by (1) water flow during a draw and (2) buoyant flows when the burner is active.
- Use data to solve current research questions, such as identifying the heat rate of the pilot light or the proportion of heat loss to ambient air through the flue as opposed to through the jacket.

3.1.1 Experimental Apparatus and Measurements

Researchers constructed an experimental apparatus to enable the collection of all necessary data. A schematic of the apparatus is shown in Figure 5.

Figure 5: Measurements and Flow Paths in Storage Tank Water Heater Test Apparatus



The experimental apparatus includes the equipment and measurements to do the following.

- Perform hot water draws at several different flow rates.
- Observe the temperature of water in the tank, enabling the study of stratification during draws and burns.
- Observe the temperature of hot gas in the flue, allowing for a study of the thermal efficiency of the unit under varying circumstances.
- Identify the hot water temperature entering and leaving the water heater.
- Characterize the functioning of the burner including rate of gas consumption by the pilot light and burner, as well as thermostat height, setpoint, and deadband.
- Monitor the electricity consumption of the fan.

3.1.2 Tested Heaters

A variety of non-condensing storage tank water heaters are available on the market. Differences among heaters include baffle design, burner design, storage tank capacity, storage tank design, and thermostat height. Data capturing the variety of non-condensing storage tank water heaters will be useful in supporting the development of more precise models. Although water heater models still tend to be generic, future models will need to capture specific effects in more detail. To support future modeling efforts, the water heaters listed in Table 1 were tested.

Table 1: Tested Non-Condensing Storage Tank Water Heaters

Manufacturer	Model Number
A. O. Smith	GNR-40-200
A. O. Smith	GPNH-40
Rheem	22V40FN

As the price of condensing storage tank water heaters gradually declines, the use of this type of water heater in homes is becoming more common. Data have been collected from one condensing storage tank water heater to help develop models for this emerging technology. Tests performed on a condensing model were identical to the tests performed on the non-condensing models. The tested condensing unit was an A. O. Smith model GPHE-50-100.

3.1.3 Experimental Protocol

Table 2 shows the behaviors to be observed and the sensors necessary to characterize and validate a simulation model for storage tank water heaters.

Table 2: Storage Tank Water Heaters: Behaviors Observed and Sensors Used

Observed Behavior	Sensors Used
Stratification in the bottom of the tank during a draw	Thermocouples inside the storage tank, cold water flow meter, cold water inlet thermocouple
Stratification in the tank when the burner is firing	Thermocouples inside the storage tank
Energy consumption of pilot light	Natural gas flow meter
Energy consumption of burner	Natural gas flow meter
Thermostat height, setpoint, and deadband	Thermocouples inside the storage tank and natural gas flow meter
Convection coefficients between the flue gas and flue wall	Thermocouples in the flue
Electricity consumption during burns and standby mode	Electric power meter
Excess air flow	CO_2 sampling in the flue

Four tests (described below) were performed to collect the necessary data for each heater: the repeated draw test, open flue UA test, plugged flue UA test, and the validation test.

3.1.3.1 Repeated Draw Test

The repeated draw test uses four draws at different flow rates. The draws are followed by burner operation until the water returns to the setpoint. The steps used in the repeated draw test are given below.

1. Start with a fully heated tank.
2. Begin a draw at 1.9 liter per minute (L/min) (0.5 gallon per minute [gal/min]).
3. Stop the draw when the burner begins firing.
4. Allow the burner to return the water to the setpoint.
5. Repeat at 4.9 L/min (1.3 gal/min), 7.57 L/min (2 gal/min), and 14.2 L/min (3.75 gal/min).

Figures 6 and 7 show examples of the data collected during the repeated draw test. The results are from a draw at 4.9 L/min (1.3 gal/min).

Figure 6: Water Temperature Measurements at Various Depths in the Tank (Measured from the Top of the Water) During a Draw

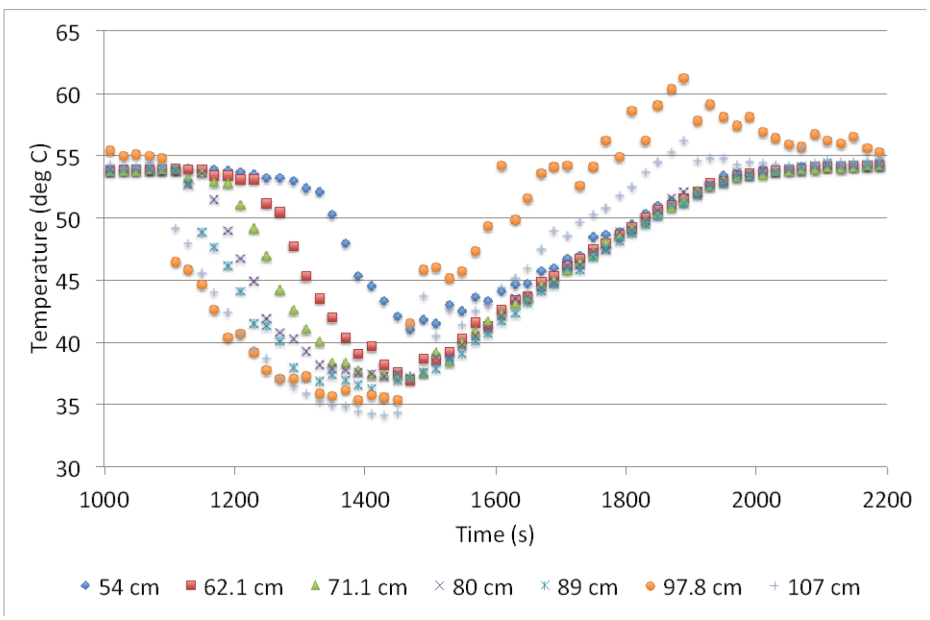
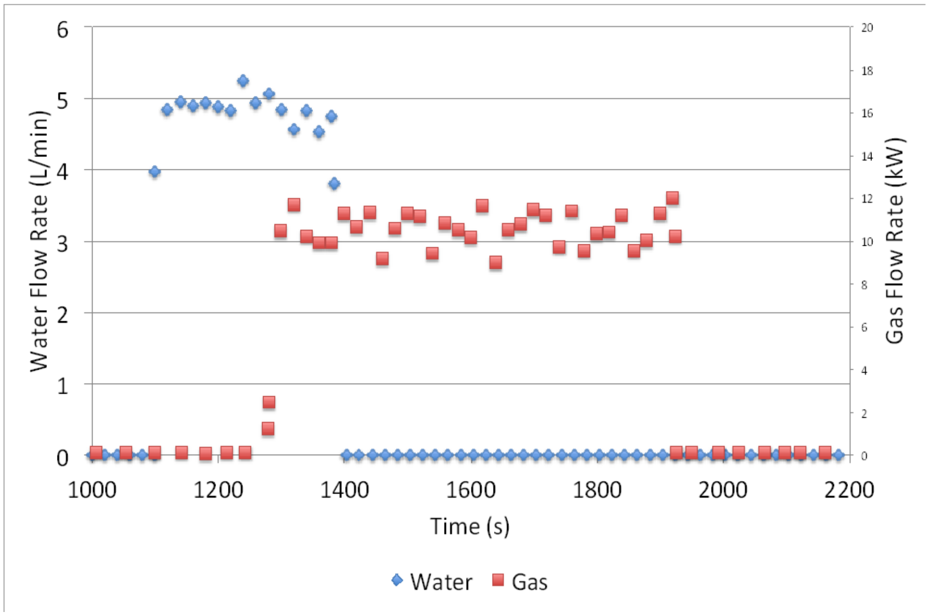


Figure 7: Water and Gas Flow Data from a Repeated Draw Test



kW = kilowatts

The data collected in the repeated draw tests can help with understanding of several storage tank water heater features.

- Temperature data during the draw (before the burner begins firing) can be used to characterize the mixing in the bottom of the tank during draws.
- For condensing water heaters, the rate of increase in water temperature during the burn can be used to identify the efficiency of the unit as a function of the stored water temperature.
- Temperature data during the burn (after the draw is stopped) can be used to understand the stratification in the tank during a burn.
- Flue temperature data during the burn are used to identify the convection coefficients between the hot gas and the base of the tank, as well as between the hot gas and the flue wall.
- Gas consumption data between burns are used to identify the energy consumption of the pilot light.
- Gas consumption during burns is used to identify the energy consumption of the burner.
- Gas consumption rate and water temperature data are used to identify when the burner turns on and off, enabling characterization of the thermostat.
- Electricity draw data are used to identify the power consumption in active and standby modes (on appropriate models).

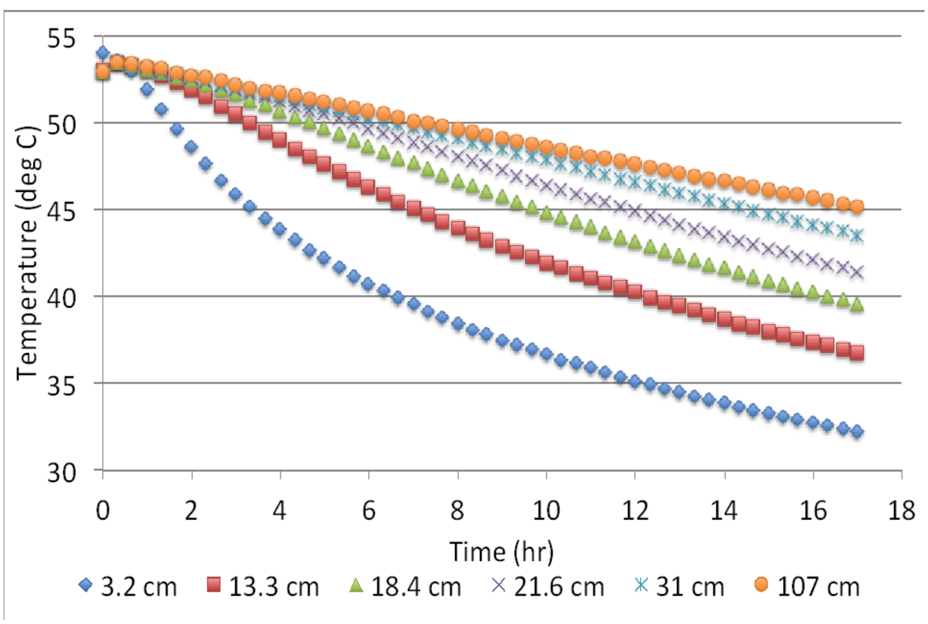
3.1.3.2 Open Flue UA Test

The open flue UA test is a simple way to identify the rate at which the storage tank loses heat to the environment. *UA* is a term commonly used to refer to the heat loss coefficient, which describes the rate at which a water heater loses heat to the ambient air. It is simply the U value (a measure of thermal conductance) times the area. The test is performed using the following steps:

1. Start with a fully heated tank.
2. Turn off the pilot light.
3. Allow the tank to cool overnight.

Figure 8 shows an example of the data collected during a open flue UA test. The names of the data series in the figure represent the depth of the thermocouples in centimeters.

Figure 8: Temperature Measurements at Various Depths in the Tank (Measured from the Top of the Water) During a UA (Open Flue) Test



The data collected in the open flue test are used to characterize the heat loss characteristics of the storage tank models. The data provide the information necessary to identify both the overall heat loss rate and the buoyancy-induced natural-convection heat flows in the stored water.

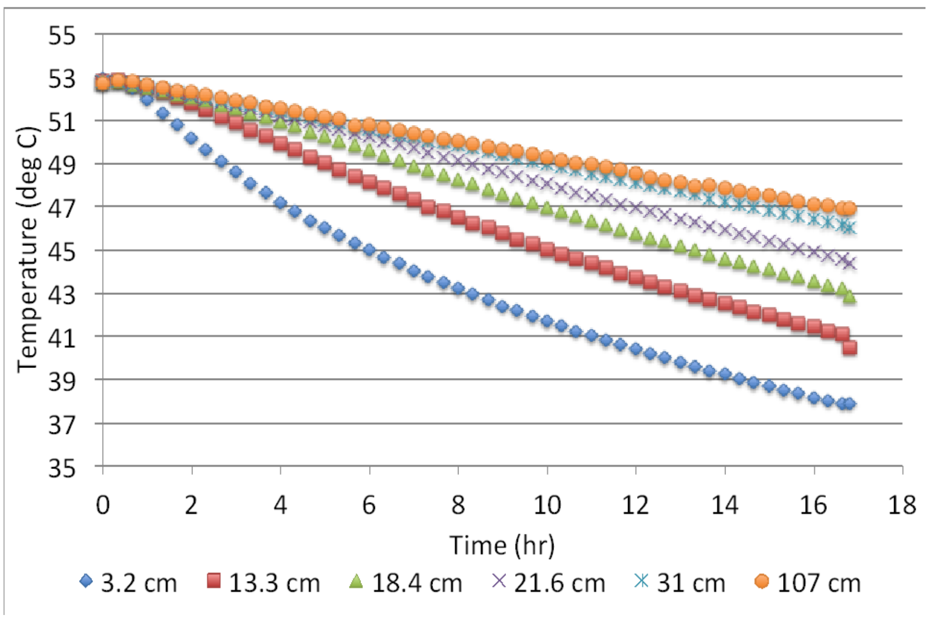
3.1.3.3 Plugged Flue UA Test

The plugged flue UA test is generally the same as the open flue UA test. The only difference is that in the plugged flue test, the flue was blocked to remove flue heat losses. The steps for performing a plugged flue test are as follows:

1. Start with a fully heated tank.
2. Turn off the pilot light.
3. Fill the flue with insulation and seal the top of the flue.
4. Allow the tank to cool overnight.

An example of the data collected during a plugged flue test can be seen in Figure 9. The data are similar to results from the open flue test, except that there was more temperature loss when the flue was not plugged.

Figure 9: Temperature Measurements at Various Depths in the Tank During a Plugged Flue UA Test



Results from the plugged flue UA test can be compared to the open flue UA results. The data are used to identify the effective UA value in both cases. By comparing results from the two tests, researchers can identify the heat loss going out through the jacket and fittings into the surrounding space, and the heat loss going through the flue and out of the building.

3.1.3.4 Validation Test

A separate draw pattern was used specifically for model validation. Because the other experimental data were used to characterize the model, they were invalid for validation, requiring an additional data set. The validation protocol was intended to provide a second data set that can be used to compare data against simulation results. The steps used in the validation test protocol are as follows:

1. *Hot water draw operation*. Perform three draws at three flow rates. Perform each draw for 20 minutes. Provide a 10-minute delay between when the burner stops firing and the next draw is started. The 20-minute draws can be used to verify mixing in the tank during the draw, as well as to check model predictions when both a draw is occurring and the burner is active. The 20-minute draw also provides different start conditions for the heat-up period after the draw ends. Draws were performed at 5.7 L/min (1.5 gal/min), 9.5 L/min (2.5 gal/min), and 11.4 L/min (3 gal/min).

2. *Heat loss to ambient conditions*. Allow the tank to sit over a weekend with the pilot light on. This test enables comparison between simulation results and experimental data when the UA losses and pilot light are the dominating heat transfers. Data from the long standby period are also used to check the thermostat behavior when there is no draw.

3. *Draws that do not trigger burns*. Perform several short draws, each lasting 30 seconds. Allow a 10-minute delay between draws. These short draws check that the simulation accurately predicts tank stratification under the circumstances. The data also are used to observe the thermostat under the circumstances.

4. *A draw that has changing flow rates*. Initiate a draw at 5.7 L/min (1.5 gal/min). After two minutes change the draw flow rate to 15.2 L/min (4 gal/min). Continue this flow rate until after the burner begins firing. Five minutes after the burner begins firing, return the flow rate to 5.7 L/min (1.5 gal/min). This test checks that the simulation model can accurately predict the mixing during a draw and burn when the flow rate changes.

3.1.4 Results of Storage Tank Water Heater Experiments

The experiments provided data for examining several behaviors exhibited by storage tank water heaters. The following sections describe various ways the data can be analyzed to understand the operation and behavior of such heaters.

3.1.4.1 Draw Mixing

Data obtained during this project can be used to study the mixing that a draw causes in the bottom of a non-condensing storage tank water heater. During a draw, cold water enters the bottom of a tank from a vertical dip tube. After exiting the tube, the water continues to flow downward toward the bottom of the tank. The water strikes the bottom of the tank, changes direction, and begins to flow back up toward the top of the tank. The details of the flow depend on the characteristics of the flow and the design of the water heater and dip tube. The draw flow rate and diameter of the dip tube determine the speed of the water that exits the pipe. The shape of the bottom of the tank determines the trajectory and scattering of the water as it begins to flow back toward the top of the tank. Both effects impact how high the water rises before becoming stagnant and following the bulk movement of the water in the tank.

Data were collected for three different non-condensing storage tank water heaters. These data can be used to characterize the effect in each heater and compare the three tanks.

Experimental results from the three tested non-condensing storage tank water heaters are shown in Figures 10, 11, and 12, which display data from the Rheem 22V40FN, A. O. Smith

GNR-40-200, and A. O. Smith GPNH-40 units, respectively. The data depict the mixing inside the three non-condensing storage tank water heaters during a water draw event (nominal 4.9 L/min flow rate in all cases).

Figure 10: Temperature Data Recorded at Several Depths in the Rheem 22V40FN Heater Showing Mixing Caused by a Draw

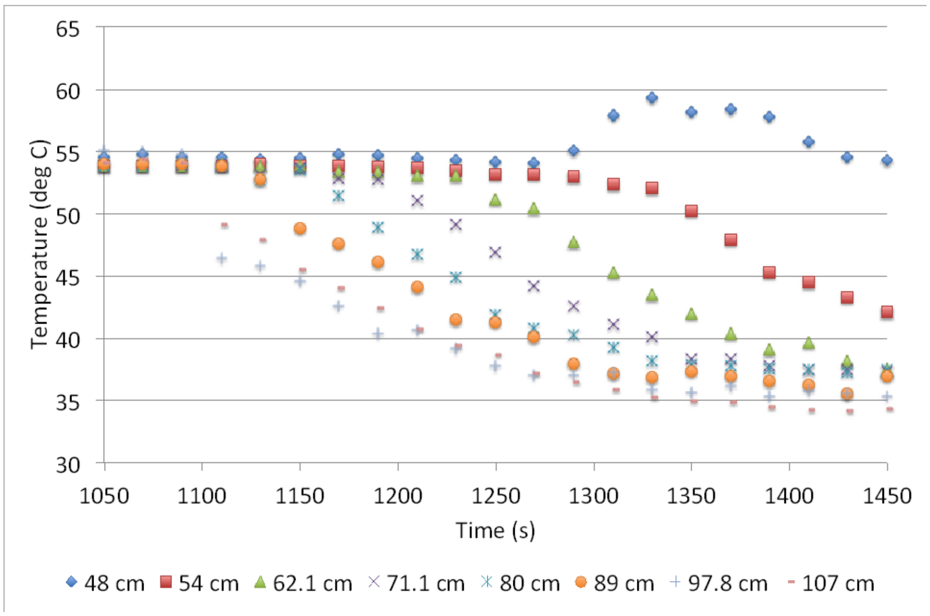


Figure 11: Temperature Data Recorded at Several Depths in the A. O. Smith GNR-40-200 Heater Showing Mixing Caused by a Draw

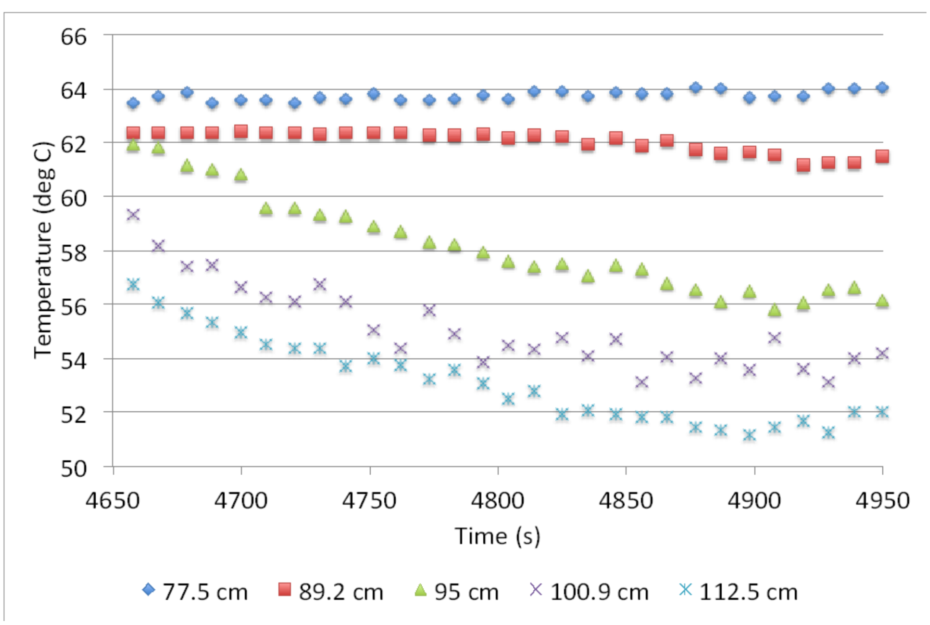
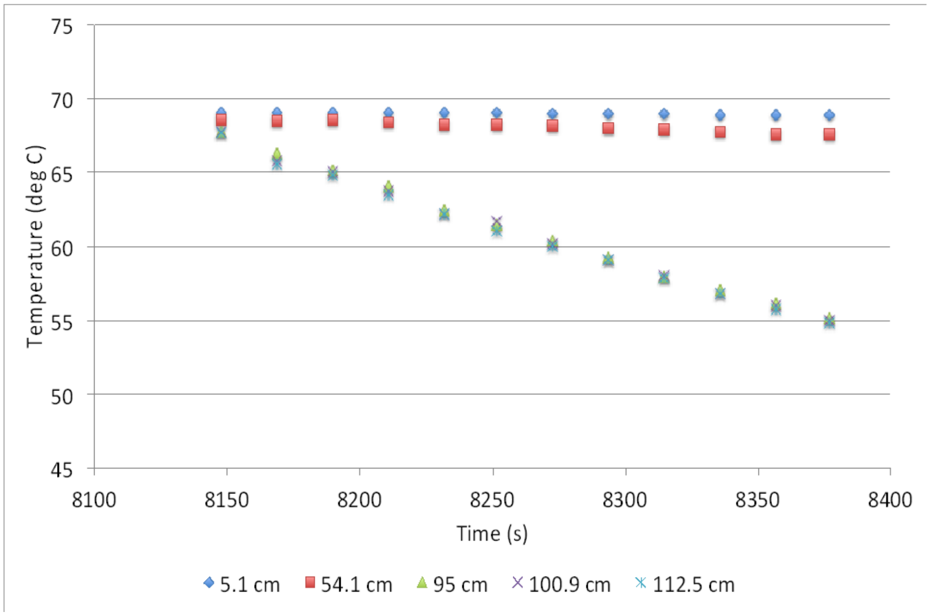


Figure 12: Temperature Data Recorded at Several Depths in the A. O. Smith GPNH-40 Heater Showing Mixing Caused by a Draw



The data displayed in Figures 10, 11, and 12 show the period between the start of the draw and the burner beginning to fire. For all three water heaters, the temperature of water stored in the tank decreased, and for all three the lower segments in the tank cooled more than the higher segments. However, the details of these effects vary dramatically from heater to heater. Some observations are listed below.

- The data in Figure 10, depicting the Rheem 22V40FN, show that the draw continued for 400 seconds (6.7 min) before the burner began to fire. Figures 11and 12, which illustrate results for the two A. O. Smith heaters, show that the burner began to fire after 250 to 300 seconds (4.17 min to 5 min). The shorter delay before firing means that the A. O. Smith heaters add more heat to the tank sooner than does the Rheem heater, and will be less likely to run out of storage capacity.

- The Rheem 22V40FN shows dramatically higher temperature loss at the bottom of the tank than either of the two A. O. Smith heaters. Part of this effect derives from a higher water flow rate: the flow rate when testing the Rheem 22V40FN was 4.9 L/min; the flow rate when testing the A. O. Smith heaters was 4.3 L/min. The higher temperature loss is also caused by the Rheem's longer delay before the burner began firing, and some is caused by the mixing characteristics of the tank itself.

- The heaters display dramatically different mixing characteristics, as illustrated by comparing Figures 10 and 12. The data in Figure 10 show that the water by a few thermocouples near the bottom of the tank is well mixed then appears to flow past the higher thermocouples in a plug-like manner. In Figure 12 the water remains well mixed throughout a larger region of the tank, and there appears to be a clearly defined stratification layer above the well-mixed layer.

3.1.4.2 Burn Mixing

Data obtained during the experiments can be used to understand and characterize the buoyant flows in a tank caused by heat transfer when the burner is active. Standard tank design includes a burner beneath the water tank, a metal base on the tank, and a metal flue rising through the center of the tank. Heater flues also have baffles that affect turbulence of the gases in the flue, and thus the heat transfer out of the flue. Design of the base, flue, and baffle and burner size impact how the water in the tank mixes during a burn.

Data were collected from three heaters. The data can help characterize the effects of burn mixing in each heater. They also can provide guidance on how to construct an expandable generic algorithm for use in a simulation model that captures the base physics and can be adjusted to model a specific tank by changing given parameters.

The data collected regarding burn mixing are shown in Figures 13, 14, and 15, which display data from the Rheem 22V40FN, A. O. Smith GNR-40-200, and A. O. Smith GPNH-40 units, respectively. The data shown in the figures depict the temperatures at various heights in the three non-condensing storage tank water heaters during a burn.

Figure 13: Temperature Data Recorded at Several Depths in the Rheem 22V40FN Heater Showing Mixing Caused by a Burn

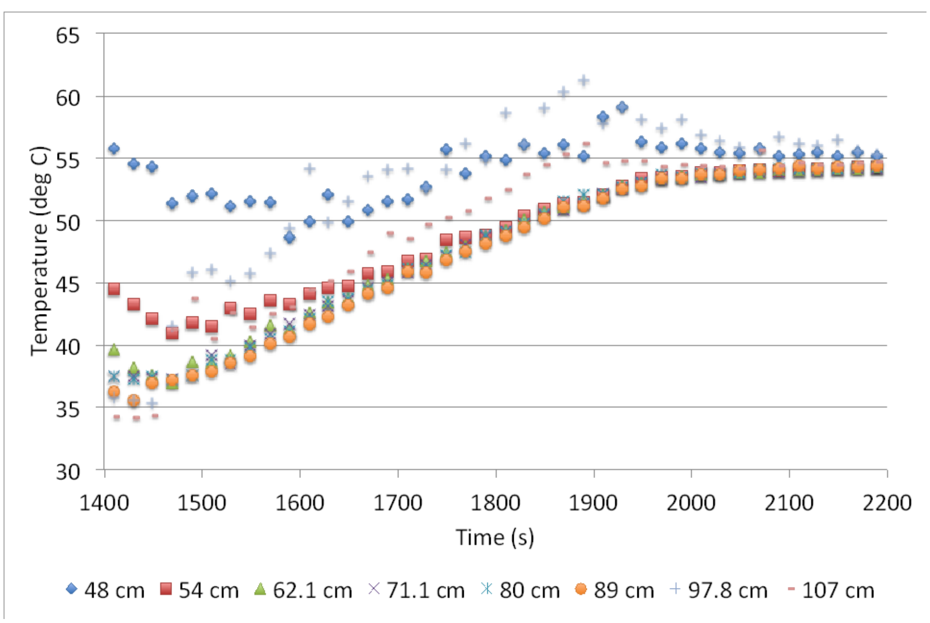


Figure 14: Temperature Data Recorded at Several Depths in the A. O. Smith GNR-40-200 Showing Mixing Caused by a Burn

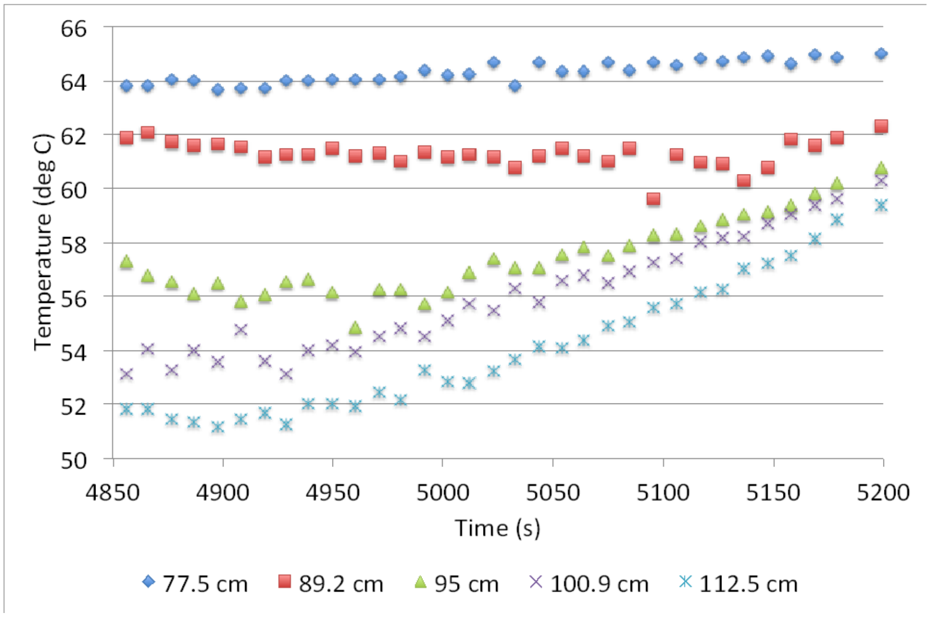
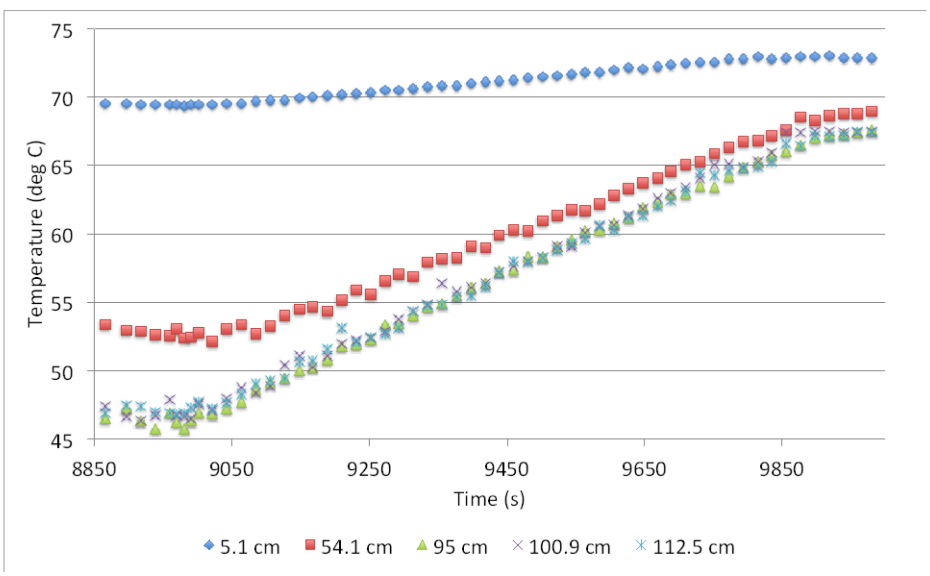


Figure 15: Temperature Data Recorded at Several Depths in the A. O. Smith GPNH-40 Showing Mixing Caused by a Burn



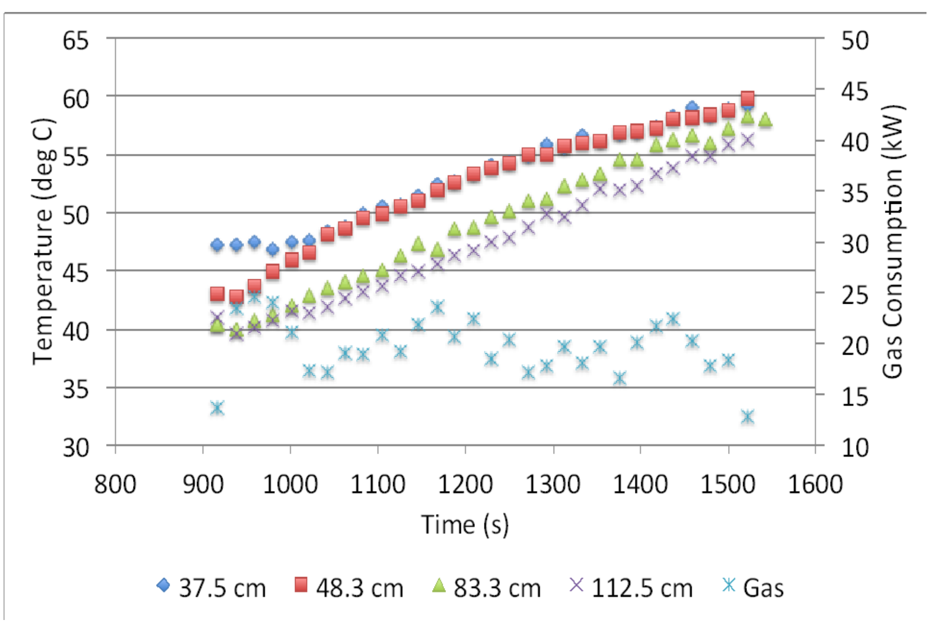
The data shown in Figures 13, 14, and 15 were collected beginning slightly before the draw stopped and continued until slightly after the burner turned off. In all three cases the temperature of all interior segments increased until the burner shut off; however, the heaters' behavior reveal several differences in the details, including the following:

- During the test the Rheem 22V40FN showed a distinct stratification layer. Temperatures measured above the stratification layer exhibited minor gain, while those below gained temperature rapidly. When the stratification layer rose to the height of a given thermocouple, the water temperature there also began to rise with the rest of the fluid beneath the layer. This behavior is also evidenced in the A. O. Smith heaters. In both the A. O. Smith GNR-40-200 and GPNH-40 the thermocouples above the stratification layer (at 77.5 and 89.2 centimeters [cm] in Figure 14 and 5.1 cm in Figure 15) showed negligible temperature rise. In most cases a clear stratification layer was preserved until the temperature of the well-mixed water below it reached that temperature.
- In the Rheem 22V40FN the temperatures recorded at the bottom of the tank spiked above the temperature of the stratification layer. This effect likely is caused by a high heat transfer rate through the base of the tank, causing high temperatures at the bottom and highly turbulent local water flows. The A. O. Smith heaters did not display this behavior, perhaps indicating that more of their heat transfer was through the flue.

3.1.4.3 Changing Thermal Efficiency

The thermal efficiency of a condensing storage tank water heater depends on the temperature of the water in the storage tank. The efficiency therefore should change over the course of a burn. As the temperature of the water in the tank increases, the efficiency of the unit will decrease. Data collected from tests on an A. O. Smith GPHE-50-100 heater can be used to determine the efficiency of the heater as a function of the temperature of the stored water. Figure 16 shows an example of the data collected.

Figure 16: Water Temperature at Various Depths and Gas Consumption Data Recorded When Testing an A. O. Smith GPHE-50-100



3.1.4.4 Gas Consumption

During the previous tests the natural gas consumption of the heaters also was monitored. Measured gas consumption enables identification of the gas consumption rate of the burner and of the pilot light gas. Both those parameters are necessary for developing an accurate simulation model. Data in Figures 17 through 19 show the gas consumption rates recorded in all three water heaters.

Figure 17: Rates of Natural Gas Consumption Recorded in the Rheem 22V40FN

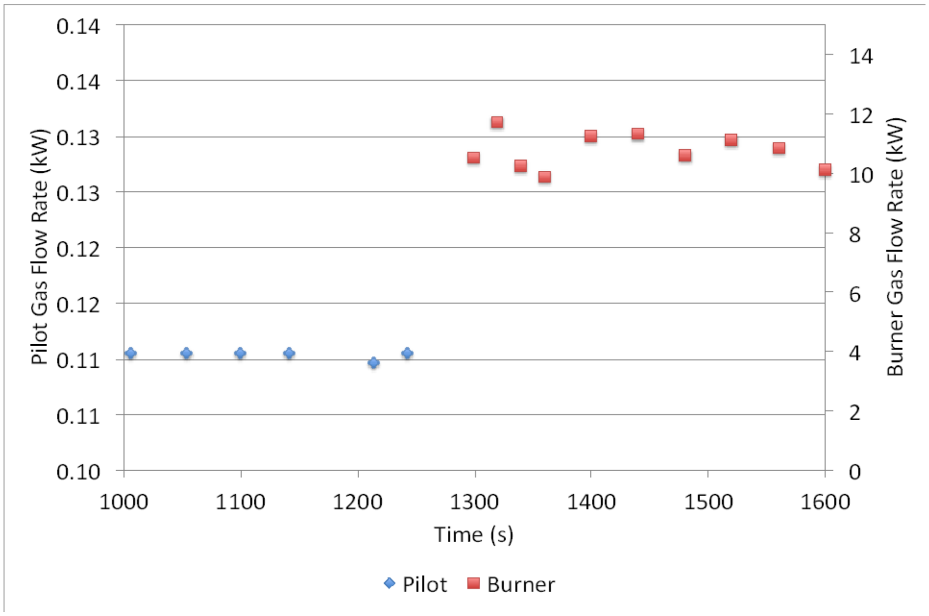


Figure 18: Rates of Natural Gas Consumption Recorded in the A. O. Smith GNR-40-200

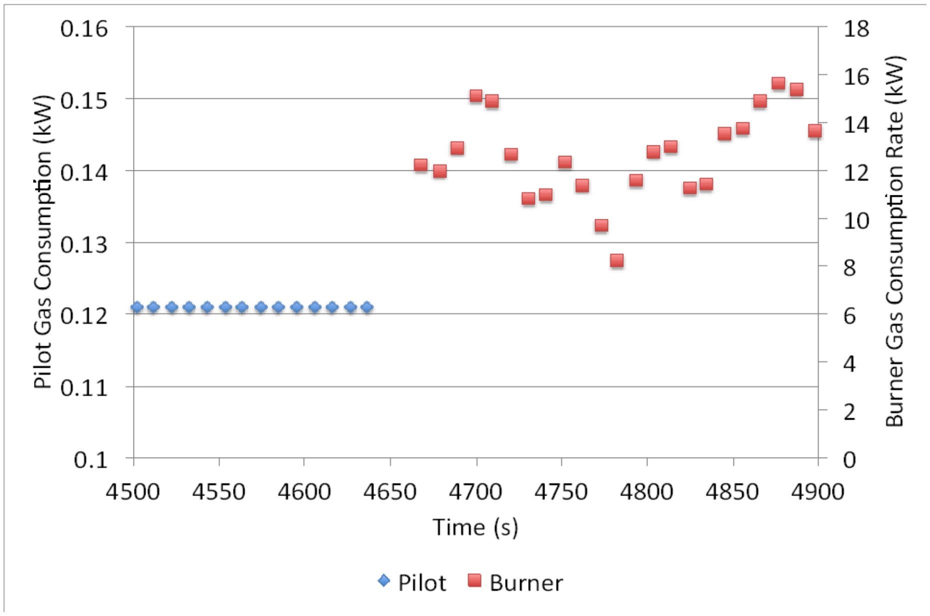
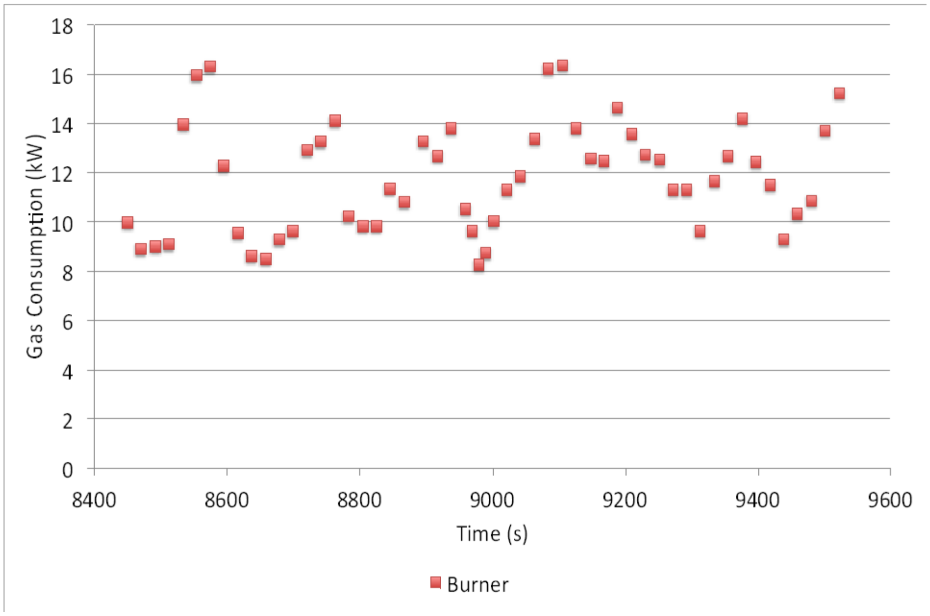


Figure 19: Rates of Natural Gas Consumption Recorded in the A. O. Smith GPNH-40-200



Where appropriate, the plots contain gas consumption data for both the pilot light and burner. The data in Figure 19 do not provide information on pilot light gas consumption because the A. O. Smith GPNH-40-200 does not have a pilot light. Figure 18 displays perfectly steady gas consumption by the pilot light because the pilot light consumption was not recorded automatically for this heater. Instead, an average value was used based on the volumetric gas flow throughout two minutes.

The experimental data on pilot light and burner gas consumption for three non-condensing storage tank water heaters can be used to generate input parameters to create simulation models for the four tested heaters. (Only two of the four tested heaters used pilot lights.)

3.1.4.5 Thermostat Control

Emulating thermostat control is an important part of modeling storage tank water heaters. It is assumed that storage tank water heaters operate with a setpoint and a deadband. The setpoint represents a central temperature, and the deadband specifies the range of operation. The heater begins firing when the thermostat measures water at the setpoint minus deadband temperature, and stops firing when the thermostat measures water at the setpoint plus deadband temperature. Correctly identifying the thermostat deadband improves predictions of water heater energy use, of energy stored in the tank, and of the temperature of the hot water leaving the tank. Thermostat setpoint is defined by a user setting, and so is not considered a constant parameter of the heater itself. The repeated mixing tests, discussed in the *Repeated Draw Test* section above, provided several data sets that could be used to identify the thermostat deadband. The resulting values are shown in Table 3.

Table 3: Identified Deadband for Each Tested Water Heater

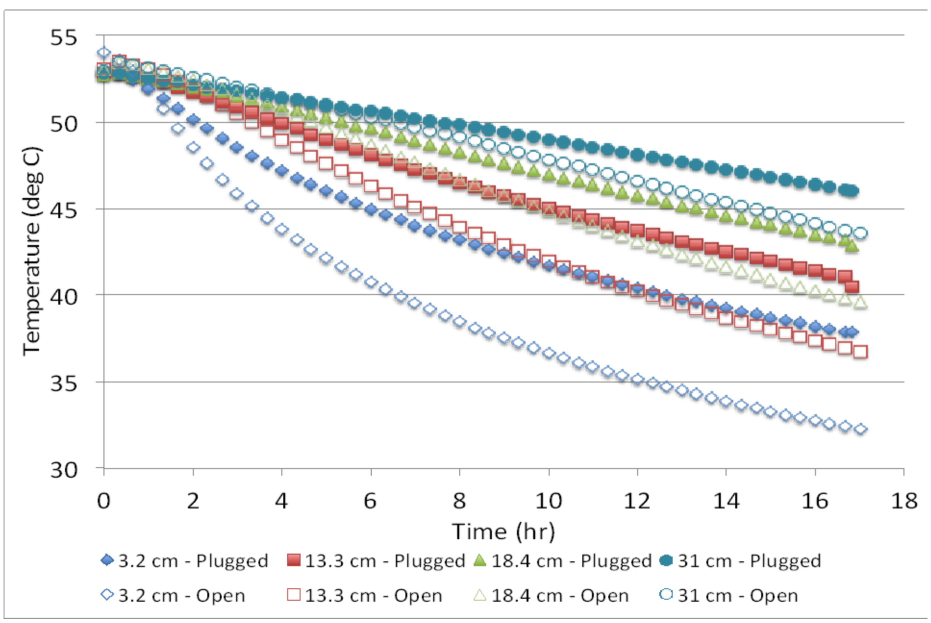
Heater Name	Deadband (±°C)/(±°F)
Rheem 22V40FN	6 / 10.8
A. O. Smith GNR-40-200	1.5 / 2.7
A. O. Smith GPNH-40	7 / 12.6
A. O. Smith GPHE-50-100	8 / 14.4

3.1.4.6 UA Tests

As described above, tests were performed to identify the rate of heat loss coefficient (UA value) for each heater. Tests were performed both with the flue open and the flue plugged. The results of those experiments enable identification of the overall UA value, the stratification behavior in the tank between draws, and the percentage of heat loss out the flue instead of through the jacket.

Figure 20 shows the data collected from tests of UA loss performed on the Rheem 22V40FN heater. Data are presented for both the open flue and plugged flue cases.

Figure 20: Temperature Measurements at Various Depths in a Rheem 22V40FN During Plugged and Open Flue UA Loss Tests



The results in Figure 20 indicate that the behavior during open and plugged flue tests is similar; with the primary difference being that heat is lost at a lower rate when the flue is plugged. This result is completely intuitive. These results provide important information when creating water heater models for use in whole-building energy simulations. The data can be used to determine how much heat is lost through the jacket and fittings of the heater, and how much through the flue. The heat leaving via the flue will leave the building out the stack, while the heat leaving via the jacket will add heat to the interior space. The collected data will enable model developers to correctly determine the two heat flow rates.

3.1.4.7 Excess Air

The concentration of carbon dioxide (CO_2) in the flue gas was measured during firing and pilot-only operation. Those measurements, combined with the gas consumption rates, will enable calculation of the excess air rates and thus the flue gas mass flow rate.

3.1.4.8 Validation Data

A separate test protocol was performed to collect a second set of data for validating models of storage tank water heaters. The validation data are available for all four heaters when modelers are ready to use them.

3.1.5 Conclusions Based on the Storage Tank Water Heater Experiments

These experiments provided data regarding several behaviors that must be included in models of storage tank water heaters. Researchers made the following observations.

- Collected data show the differences in mixing caused by hot water draws at different flow rates and in different water heaters. These data can be used to understand the effects of mixing, develop an algorithm to model the effects, and identify the parameters needed to model the effects in each heater. The data can also be used to validate the model.
- Data present similar information for the mixing effects caused by a burn. They can be used to understand what is occurring in the heater, create a model algorithm to simulate the processes, identify the parameters needed to describe each heater, and validate the model.
- Gas consumption data can be used to characterize the pilot lights and burners of each tested heater.
- Collected data enable researchers to understand and characterize the heat loss and stratification effects during standby periods. The characterization can be performed for the entire heat loss rate, as well as for the heat loss rate through the jacket, enabling the development of models for use in whole-building simulations.

3.2 Non-Condensing Tankless Water Heaters

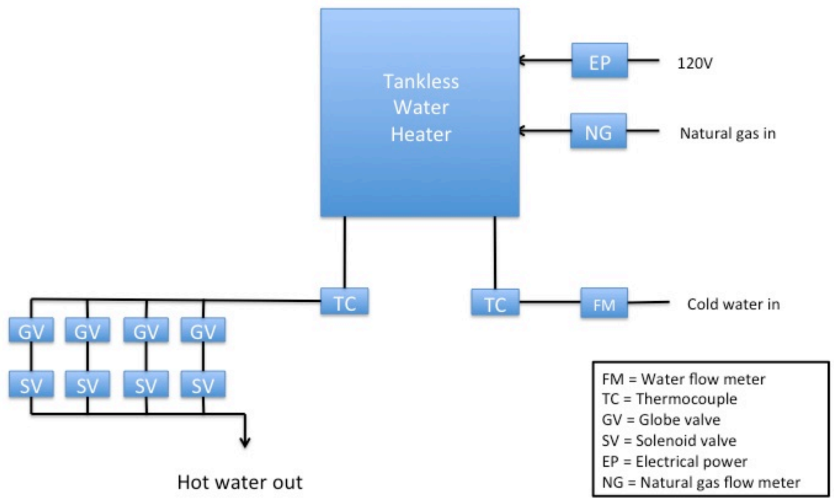
Non-condensing tankless water heaters represent a rapidly growing segment of the residential water heating market. They provide improved efficiency and potential for limitless hot water, and increasing in market share (Ryan et al. 2010).

Experimental data was collected on the operation of non-condensing tankless water heaters. These data will support future work developing simulation models.

3.2.1 Experimental Apparatus and Measurements

Researchers modified the experimental apparatus storage tank water heaters to test non-condensing tankless water heaters. Figure 21 shows a schematic of the modified apparatus.

Figure 21: Measurements and Flow Paths in the Tankless Water Heater Test Apparatus



The experimental apparatus includes all of the measurements and equipment needed to:

- Perform hot water draws at several different flow rates.
- Identify the temperature of the water entering and leaving the water heater.
- Identify the UA value of the unit, as well as the thermal capacitance based on the energy remaining in the heat exchanger under various conditions.
- Monitor the natural gas consumption of the burner and identify the steady-state efficiency, as well as the proportional, integral, and derivative values (PID) used to program common feedback controllers. These constants can be used to emulate the controller.
- Monitor the electricity consumption of the fan, igniter, and controls.
- Identify the control logic, including response time to changing water flows and purge time before and after a draw.

3.2.2 Tested Heater

One heater was available for testing, a Rheem 84-DVLN. Table 4 presents the specifications from the manufacturer for this unit.

Table 4: Manufacturer Specifications for the Tested Rheem 84-DVLN

Parameter	Unit	Value
Efficiency	percent	82
Energy Factor		0.82
Minimum Flow Rate	L/s gal/min	0.016 0.254
Activation Flow Rate	L/s gal/min	0.025 0.4
Minimum Gas Input	kW kBtu/hr	3.2 10.9
Maximum Gas Input	kW kBtu/hr	58.3 199

3.2.3 Experimental Protocol

Several tests were performed on the tankless heater to collect data necessary to create a valid simulation model of the heater. Data were collected for use in examining several behaviors exhibited by tankless heaters. Table 5 shows the observed behaviors and measurements used.

Table 5: Observed Behaviors and Measurements Collected for Tankless Water Heater

Observed Behavior	Measurements Collected
Steady-state efficiency of the burner	Inlet temperature, outlet temperature, water flow rate, and gas consumption during a draw
Thermal capacitance of the heater	Inlet temperature, outlet temperature, and water flow rate during a very slow draw after a hot water draw. The slow water draw is low enough that the burner does not fire.
UA value of the heat exchanger	Inlet temperature, outlet temperature, ambient temperature, and water flow rate during a very slow draw for a specified amount of time after a draw ended
Time delay after a draw starts before the burner begins firing	Water flow rate, natural gas flow rate, and electricity consumption
Time delay after a draw ends before the fan stops purging flue gases	Natural gas consumption and electricity consumption
Electric power consumption in standby	Electricity consumption
Electric power consumption during operation	Electricity consumption
Excess air flow	CO_2 sampling in the flue

All of the above-listed behaviors were identified using a single test. A second test was performed to collect data for use in validation. The two tests are described in the following two sections, *Characterization Data* and *Validation Data*.

3.2.3.1 Characterization Data

A test protocol was followed in obtaining data for characterizing the tankless water heater. The characterization protocol was designed to obtain the data necessary to create a tankless water heater model that could replicate the observed behaviors. The following data were of interest.

- Steady-state burn data to identify the steady-state efficiency and PID constants. The burns were performed at three flow rates and three set temperatures to determine whether those changes affected the efficiency or controller behavior.
- Data from periods of low flow after a burn to identify the thermal capacitance (low flow used to push hot water out of the unit and identify the energy stored in the heat exchanger). These data were collected at the end of each steady-state burn, creating several data sets for study.
- Data from periods having no flow after burns, followed by low flow, to identify the UA value (low flow used to push hot water out of the unit and identify the energy remaining in the heat exchanger). This test was performed using delay periods of 1, 5, 10, 30, and 77 minutes.
- Data from several draws having short delays between them to identify the delay between a draw starting and the burner firing. Repeat draws were performed with delays of 5, 10, 40, 80, 160, and 320 seconds.

3.2.3.2 Validation Data

The validation test protocol was followed again to generate a second set of data that also demonstrate various behaviors of the tankless water heater. Those conditions were input to a model so that simulation results could be compared to the experimental data to determine how well the simulation model emulated the actual behavior of the tankless water heater. The steps of the validation test protocol, and the reasons they were included, are outlined below.

- A draw is performed at 5.7 L/min (1.5 gal/min) flow rate and 46 °C (114.8 °F) set temperature. Because neither this flow rate nor setpoint were used in the characterization test, this draw tests the ability of the model to emulate controller reaction during the transient periods between heater operation and steady-state efficiency.
- A sudden change is made to a draw at 9.5 L/min (2.5 gal/min) and 46 °C (114.8 °F) setpoint. Again, this flow rate was not used in the characterization profile, providing the opportunity to again test the identification of steady-state efficiency. The sudden change in flow rate also tests the model's ability to predict how the heater responds to changes of flow rate during a draw.

- Draws are performed at different flow rates with the 46 °C (114.8 °F) setpoint. The changes in flow rate are all sudden and with no delay between them. This procedure provides several sets of data to study water heater behavior during the sudden change in flow rate.
- The changing flow rate draw is followed by a delay of 15 minutes. This delay tests the UA losses at a different time delay than in the characterization test.
- The 15-minute delay is followed by a draw at 13.2 L/min (3.5 gal/min) flow and 41 °C (105.8 °F) setpoint. This procedure provides another combination of flow rate and setpoint that was not tested in the characterization test. In addition, the draw is started with hot water still in the heat exchanger, unlike in the characterization test, where the water was pushed out using a slow draw. This difference enables testing the model's ability to predict the cold water sandwich effect.
- The draw is stopped and followed by a 90-minute delay.
- Then a draw of 9.5 L/min (2.5 gal/min) flow rate and 41 °C (105.8 °F) setpoint is performed. This procedure provides a second draw at 41 °C (105.8 °F) setpoint for purposes of examining the steady-state efficiency and control logic capabilities of the simulation model.
- The final draw is followed by a 45-minute decay period. The decay period is followed by an extremely low water flow rate that pushes the hot water out of the heat exchanger. This procedure provides an accurate check of the heat lost to ambient conditions between draws.

3.2.4 Results of Non-Condensing Tankless Water Heater Experiments

The collected data can be used to study the operation of tankless water heaters. The data from the characterization test can be used to identify parameters that describe the Rheem 84-DVLN non-condensing tankless water heater. The derived parameters then can be used in a simulation to validate the model against the validation test protocol data.

3.2.4.1 Characterization Test

The characterization test consisted of several phases that attempted to identify various parameters. The data for each phase will be presented separately due to the large volume of data collected during the characterization test.

3.2.4.2 Repeated Draws

The first phase of the characterization test consisted of several hot water draws at varying flow rates and set temperatures. The data collected in this phase can be used in the following ways:

- The gas consumption, water flow rate, and temperature difference through the heater during a draw can be used to identify the steady-state efficiency. Because draws were performed at various flow rates and set temperatures, the data allow analysis of the efficiency under a range of conditions.

- The rate of temperature decay observed in the outlet water after the draw flow is reduced indicates the capacitance of the heat exchanger. The capacitance can be identified by adjusting the capacitance value in the simulation until the outlet temperature in the simulation results matches the experimental data.
- The gas consumption data during transient portions at the start of a draw can be used to identify the PID constants that characterize the heater. The P, I, and D values in the simulation model can be adjusted until the rate of natural gas consumption during the transient period closely matches the experimental data.

Figures 22 and 23 present data collected during the first phase of the characterization test. The figures are showing a reduced subset of the data. To render the plots readable, the displayed data have 120 seconds (s) between each data point, while collected data have only 1 s between each point.

Figure 22: Temperature Difference Across the Heater and Water Flow Rate Measured During the Repeated Draws Part of the Characterization Test

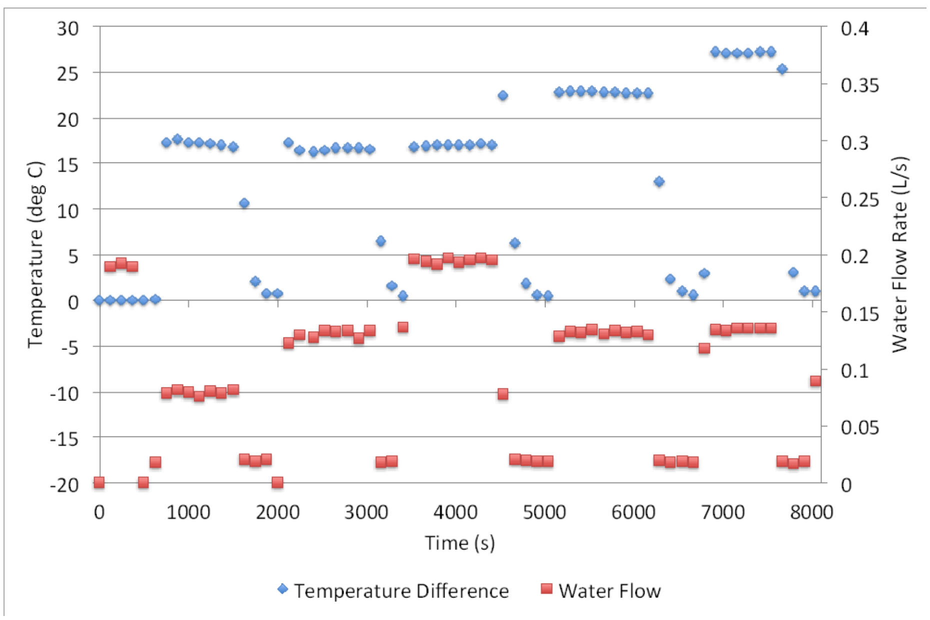


Figure 23: Natural Gas Consumption Measured During the Repeated Draws Part of the Characterization Test

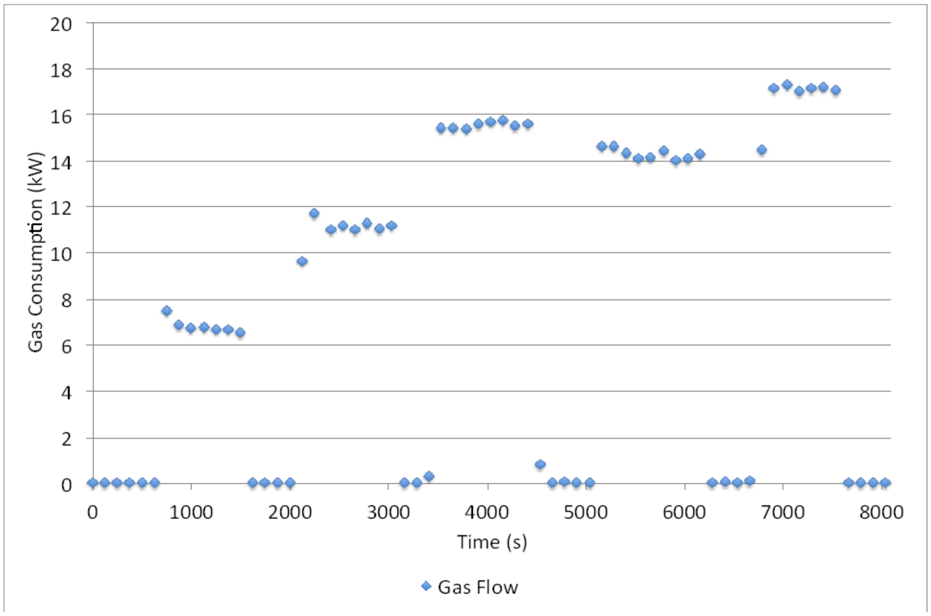


Figure 22 shows the temperature difference across the heater and the water flow rate during the test. The test involves six periods of water flow. The first, occurring at 100 seconds (1.7 min), is a purge period used to push water out of the system and replace it with water at mains water temperature. The remaining water flows are draws performed at 0.07 (0.018), 0.13 (0.034), and 0.2 (0.05) L/s (gal/s). Between draws is a short period of 0.02 (0.005) L/s (gal/s) water flow. This low-flow period between draws was used to push hot water out of the heat exchanger, allowing identification of the remaining heat in the heat exchanger and thus the capacitance of the unit.

As Figure 22 also shows, the temperature difference between the outlet and inlet temperatures differed from draw to draw. The temperature difference, in this case, is defined as the outlet temperature minus the inlet temperature. Changes in the temperature difference indicate an increase in heater set temperature.

Figure 23 shows the rate of natural gas consumption during the repeated draws part of the characterization test. Because the resolution of the displayed data is low, they display only the steady-state consumption. The collected data, which are higher resolution, provide information on how the heater behaves during the transient period at the start of the draw. Figure 24 shows the higher-resolution data representing the rate of natural gas flow for the first 40 seconds of a draw.

Figure 24: Higher-Resolution Gas Consumption Data from the First 40 s of a Draw

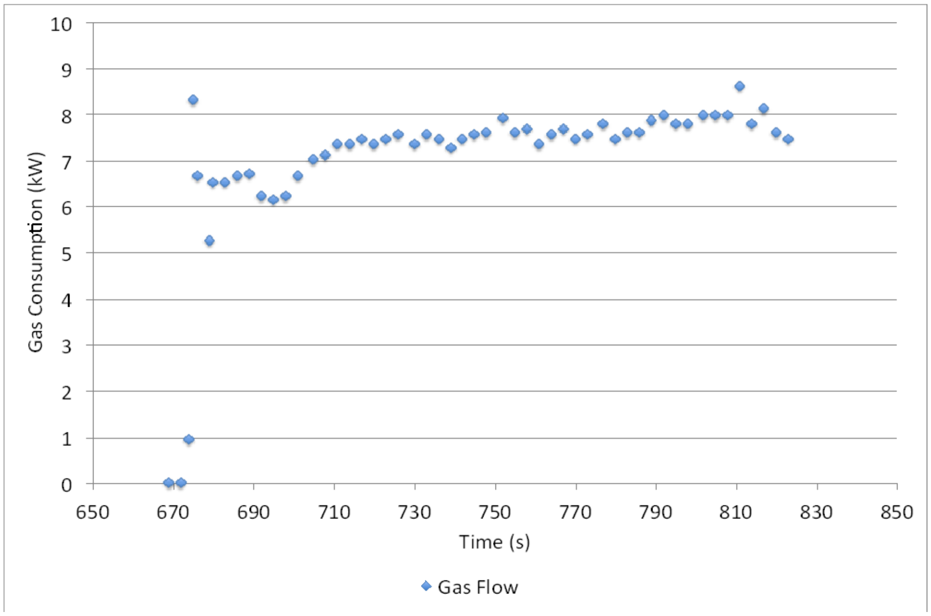
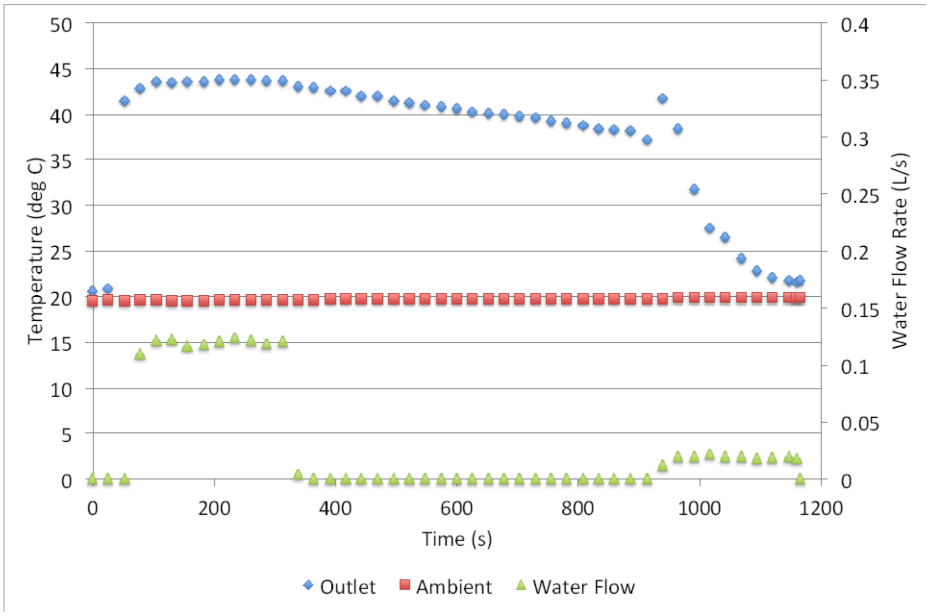


Figure 24 shows natural gas consumption data collected every 3 seconds during the transient part of a draw. These data can be used to identify the P, I, and D constants used by the heater to control the gas consumption at the start of a draw.

3.2.4.3 UA Decay Tests

The second phase of the characterization test consisted of several short burns, followed by delays of varying lengths between draws. At the end of a decay period, a draw at a low flow rate was used to push the remaining energy out of the heat exchanger. The capacitance of the heat exchanger had been identified in the repeated draws test. The heat loss coefficient (UA value) of the heater is the sole remaining variable that affects the energy in the heat exchanger at the end of the decay periods. As a **result**, the delays between draws can be used to identify the UA value of the heater. Figure 25 presents an example data set from the UA decay phase of the test. Only some of the data are shown because the copious amount of data renders a plot displaying all of the data unreadable.

Figure 25: Temperature and Flow Rate Measured During a 10-Minute UA Decay Test



The data shown in Figure 25 depict a complete UA decay test. An initial period of no water flow separates this test from the previous test. At 80 seconds (1.3 min), the water flow rate is increased to 0.12 L/s (0.03 gal/s) for 240 seconds (4 min). The increased flow rate initiates a burn that increases the heat exchanger temperature. After the draw, water flow was stopped for 600 seconds (10 min), allowing the heat exchanger to lose heat to the ambient conditions. After the 600 second decay, the water flow rate was increased to 0.02 L/s (0.005 gal/s), pushing the remaining thermal energy out of the heat exchanger. These data can be used to identify how much heat was stored in the heat exchanger at the end of the draw. Thus the UA value can be identified. After the energy is removed from the heat exchanger, the flow rate is stopped in preparation for the next test.

This test protocol was performed using delays of 1, 5, 10, 30, and 77 minutes.

3.2.4.4 Firing Delay Tests

The third phase of the characterization test focused on the delay between the beginning of the water draw and the burner firing. Tankless water heaters do not react instantly at the beginning of a draw; it often takes a few seconds for their controls to recognize a draw, the fan to purge the air in the combustion chamber, and the burner to begin firing. Those delays can result in erratic outlet water temperature. Situations will arise when hot water remaining in the heat exchanger is pushed out by the draw, to be replaced by cold water entering the heat exchanger. Eventually the heater reaches full operation and outputs hot water. The extent of the temperature fluctuations is affected by the delay period between the draw starting and the heater firing. The tests brought the heater to steady-state operation, and then stopped flow. After a pre-determined period, the water flow was restarted to begin the next draw. Figure 26 shows sample experimental data.

Figure 26: Gas and Water Flow Rates Recorded During Firing Delay Test

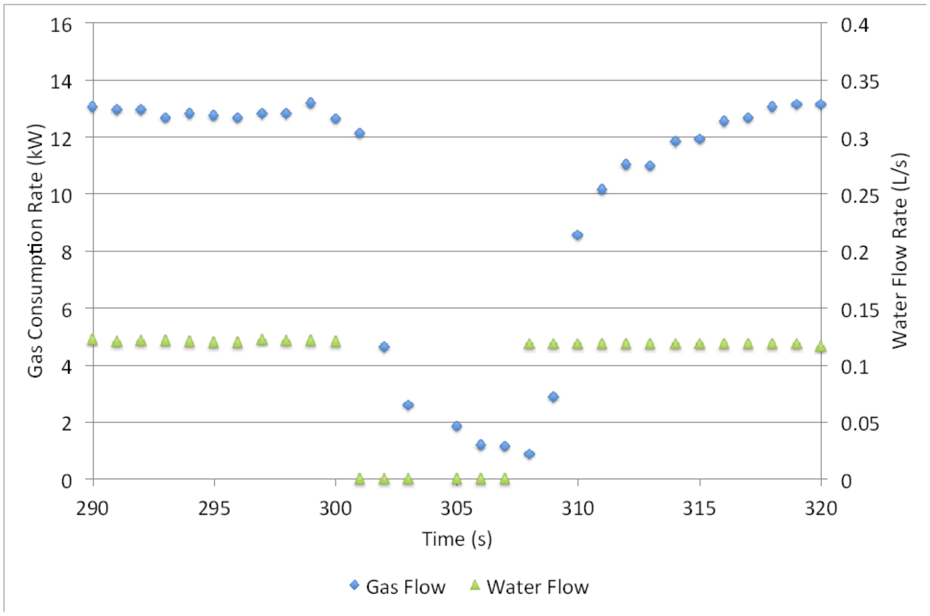
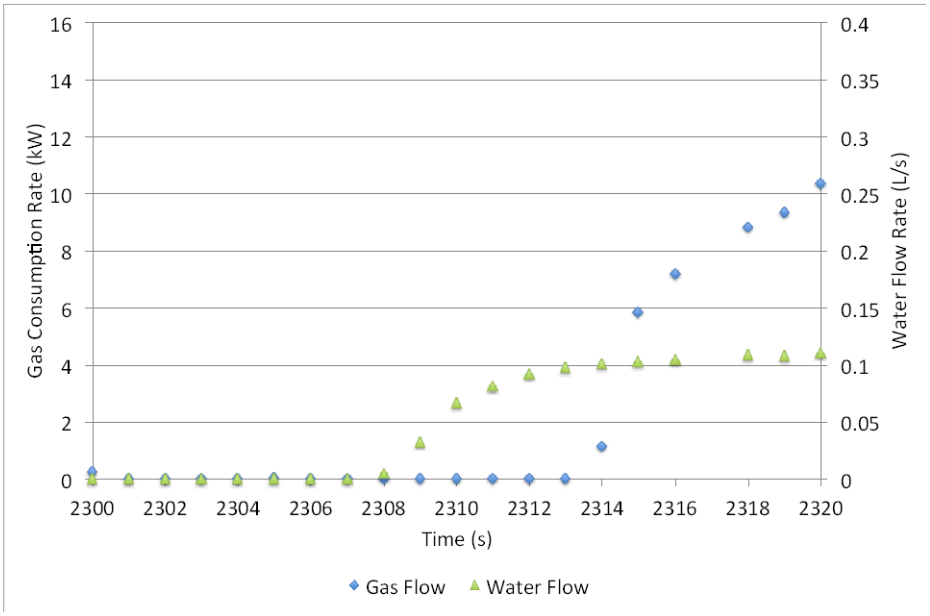


Figure 26 illustrates that the reaction of the heater is delayed on both ends of the draw. The water flow was stopped at 300 seconds (5 min), but the natural gas remains at steady-state operation until 301 seconds (5 min, 1 second). This delay increases the temperature of the heat exchanger and affects the results of the UA decay tests. There is also a delay of approximately 1 second when the water flow is resumed at 308 seconds. The natural gas flow does not begin to increase until 309 seconds.

Tests such as the one represented in Figure 26 were performed with delay lengths of 5, 10, 20, 40, 80, 160, and 320 seconds. Figure 26 shows the results from a test having a 5-second delay. Figure 27 shows the increase in the rates of water and natural gas flow at the end of the 320-second delay.

Figure 27: Gas and Water Flow Rates Measured at the End of a 5.3 Min Delay



The data in Figure 27 show the delay between the beginning of the water flow and the heater beginning to burn natural gas. The displayed data begin near the end of the 320-second delay. At 2,309 seconds the water draw begins, as shown by the increase in water flow rate on the plot. The natural gas flow rate remains at zero until 2,314 seconds, when the flow rate beings to increase. These data indicate approximately a 5-second delay between when the water begins to flow and the heater begins burning gas.

Data are available for several different delays between draws. The different delays can be used to study and characterize the control logic of the heater. The characterized values can then be entered into a simulation model to perform simulations emulating the tested Rheem 84-DVLN.

3.2.4.5 Excess Air

The concentration of CO_2 in the flue gas was measured during water heater operation. Those measurements, combined with the gas consumption rates, will enable the excess air rates and thus the flue gas mass flow rate to be calculated under different operating conditions.

3.2.4.6 Validation Data

As discussed above, a separate test protocol was performed to collect a second set of data that can be used to validate models of non-condensing storage tank water heater. The validation data are available when modelers are ready to use it.

3.2.5 Uses for Non-Condensing Tankless Water Heater Experimental Data

Data collected for this project allow the study of several behaviors of tankless water heaters. The collected data can be used in the following ways:

- The data collected through this task can be used to study the effect of draw flow rate and setpoint temperature on the steady-state efficiency of the tested tankless heater. Draws were performed at various flow rates with various set temperatures, enabling a user to identify the efficiency under any of the given conditions.
- Data are available to identify the capacitance of the heat exchanger at both low and high flow rates. The two different data sets allow the researcher to identify the capacitance of the Rheem 84-DVLN heater, as well as to determine the best method for identifying the capacitance of the heater.
- The long decay periods between draws enable the researcher to identify the UA value of the tested heater. The heat stored in the heat exchanger at the end of the decays will be reduced based on when the draw stopped, because of thermal losses to the surroundings. These effects can be studied and characterized using the collected data.
- The collected data can be used to understand and describe the control logic in the heater. The delay period between a draw starting and the gas burner beginning to fire can be identified based on data from the firing delay tests. The PID constants can be identified from either the repeated draw tests or the firing delay tests.

Validation data can serve as a companion to characterization data. Although most of the data collected through the experiments described in this chapter pertain to the water heater, and can be used for characterization, the validation data provide a means of comparison. Any data output from a simulation model must match the validation data in order for the model to be considered valid.

Chapter 4:
Water Heater Simulation Models

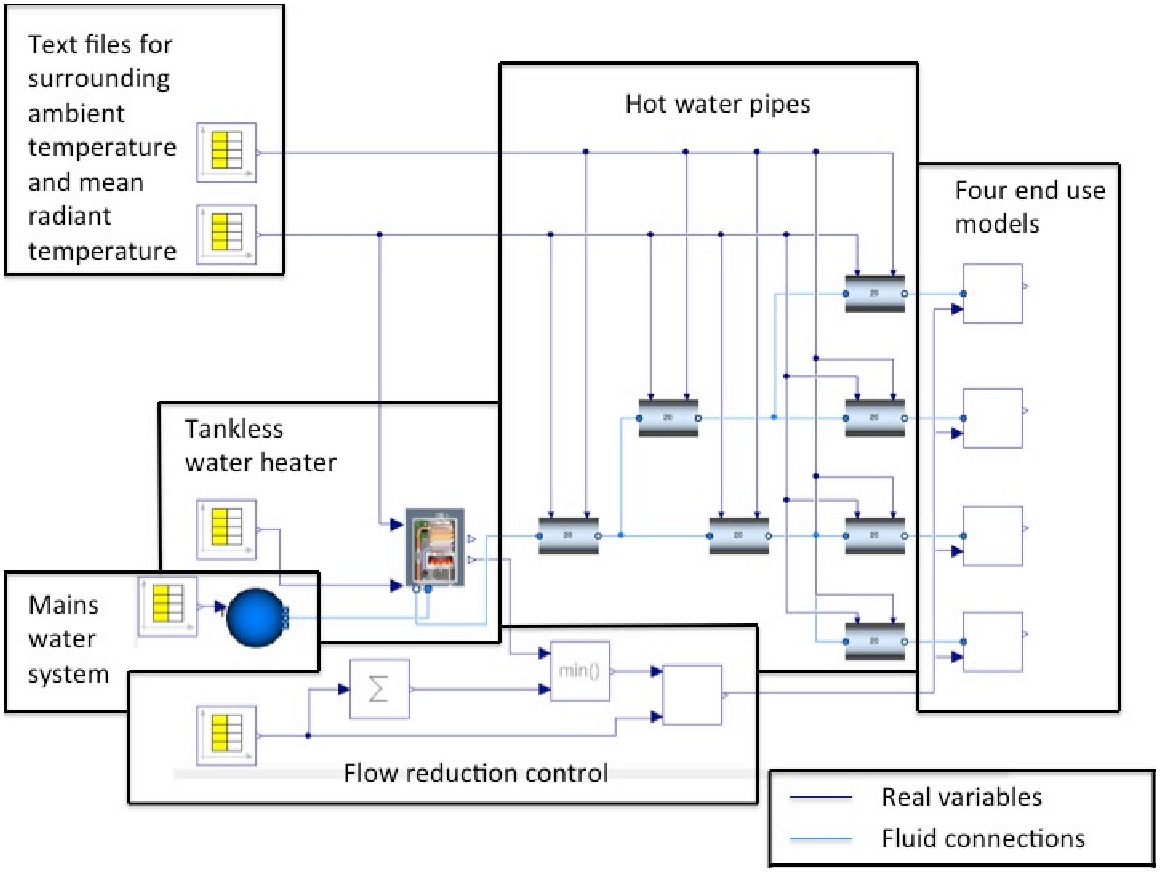
The LBNL research team used the Modelica language to create computer models for simulating residential hot water systems. These simulation tools will be included in the LBNL Buildings Library for use in whole-building energy simulations (Wetter et al. 2011). Creating models for water heaters in Modelica, using the Dymola development studio, allows easy connections with the hot water distribution system models (Tiller 2001; Dynasim AB 2009).

The water heater models also could be used to inform the Title 24 revision process. Additionally, they are capable of being expanded to accommodate new technologies.

The LBNL Water Heating library is a collection of simulation models for residential hot water systems. The library contains models of several types of water heaters, as well as models used to simulate the distribution pipes and end uses. The library enables users to quickly assemble a model of a residential water heating system by dragging and dropping system components. This chapter describes the top-level models of two types of water heaters—non-condensing storage tank water heaters and a non-condensing tankless water heater—as well as the component models that contribute to the top-level models.

Figures 28 and 29 show example simulations of residential water heating systems that can be performed using the Water Heating library. Figure 28 depicts a simulation in which a non-condensing tankless water heater provides hot water for a residence.

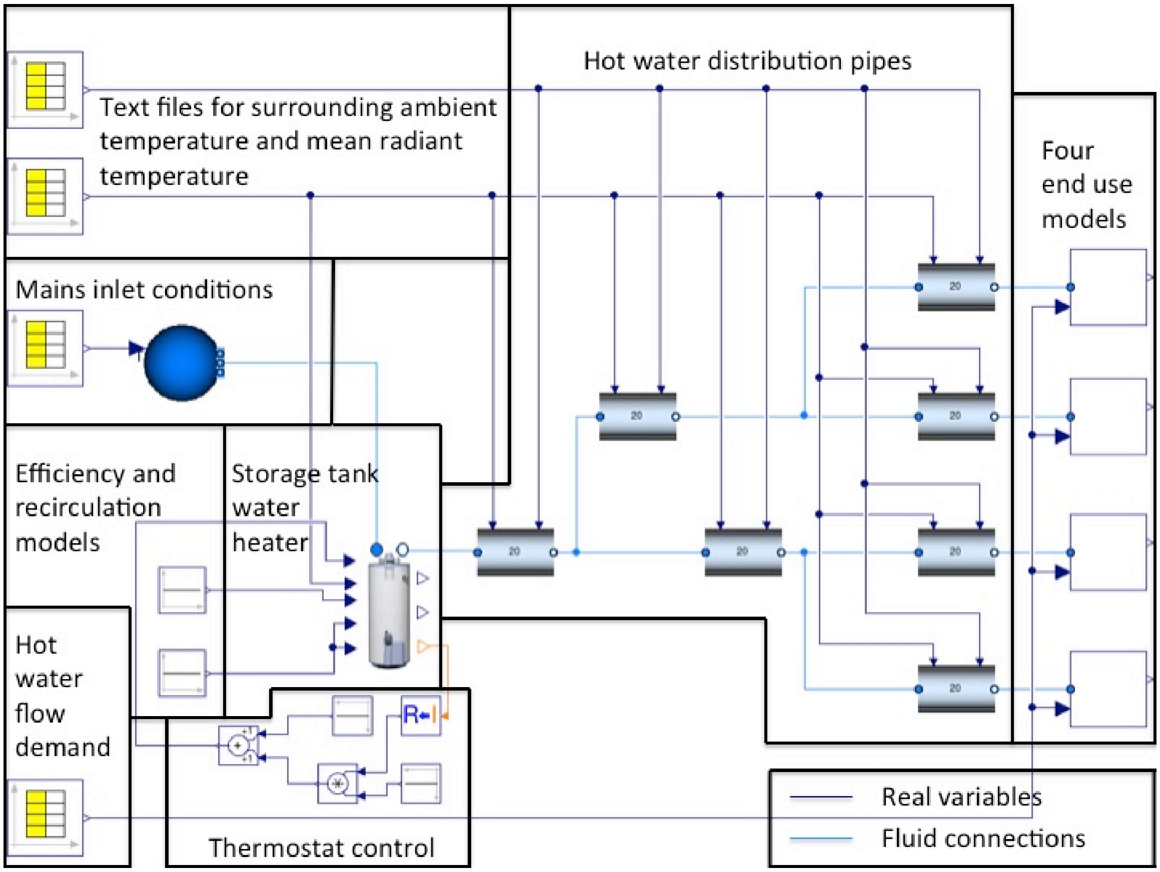
Figure 28: Residential Water Heating System Including a Non-Condensing Tankless Water Heater and Four End Uses



The models on the left side of the schematic provide information about current operating circumstances. Those models read data files to provide information on the surrounding air temperature, surrounding mean radiant temperature, and mains water temperature. All the models provide information to the tankless water heater model. The tankless water heater model then identifies the temperature of water entering the hot water pipes. The model estimates the volumetric, thermal mass, and heat loss effects as the water passes through the pipes to the end uses. At the right-hand side of the schematic, the four end-use models provide outlets that can be used to simulate sinks, showers, and other hot water end uses.

Figure 29 shows a schematic of a storage tank water heater.

Figure 29: Simulation of Residential Hot Water System Including a Non-Condensing Storage Tank Water Heater and Four End Uses



The model of the storage tank water heater requires a few external models that describe the thermostat control, as well as the efficiency of the unit and connections to a recirculation loop. The efficiency of the model is described in a component model and entered into the tank model as a time-varying input. The ability to use a time-varying efficiency model is necessary when developing models of condensing storage tank water heaters. The recirculation loop ports can be used to connect the storage tank water heater to a solar thermal collector, or to connect to a tankless heater in a hybrid water heater model. The external thermostat control model reads a binary on/off signal from the heater model and identifies the current heat rate. The pilot light heat rate is used if the burner sends an "off" signal. If the burner sends an "on" signal, the burner heat rate is used. Having the heat rate identified externally to the tank allows for connections with tankless water heaters for creating hybrid water heater models.

4.1 Storage Tank Water Heater Model

A model library must include models of non-condensing storage tank water heaters because this type of equipment is currently the predominant form of water heating in the United States (Ryan et al. 2010). Lawrence Berkeley National Laboratory developed a top-level model of a non-condensing storage tank water heater, as well as several component models that can be used or modified to develop custom simulation models. The component models are designed to allow expansion for future work on condensing and hybrid water heater models.

4.1.1 Using the Model

Users specify parameters in the top-level model to describe a specific water heater. The available parameters are described briefly in Table 6. Several of the parameters, which are specific to one of the component models, are discussed below.

Table 6: Parameters in the Non-Condensing Gas Storage Tank Water Heater Model

Medium	The fluid being heated. By default this references a model for water in the LBNL Buildings Library.
nSeg	Number of segments in the simulation
T_Initial	Initial temperature of fluid in each segment of the tank
perInA	A coefficient in the polynomial describing the water flow entering each segment in the tank during a draw. More detailed information is provided in the section below that describes draHeaTra.
perInB	B coefficient in the polynomial describing the water flow entering each segment in the tank during a draw. More detailed information is provided in the section below that describes draHeaTra.
perInC	C coefficient in the polynomial describing the water flow entering each segment in the tank during a draw. More detailed information is provided in the section below that describes draHeaTra.
timDelA	A coefficient in the polynomial describing the time delay before mains water begins to enter each segment in the tank during a draw. More detailed information is provided in the section below that describes draHeaTra.
timDelB	B coefficient in the polynomial describing the time delay before mains water begins to enter each segment in the tank during a draw. More detailed information is provided in the section below that describes draHeaTra.
timDelC	C coefficient in the polynomial describing the time delay before mains water begins to enter each segment in the tank during a draw. More detailed information is provided in the section below that describes draHeaTra.
TStatHeight	Segment of the tank in which the thermostat is located
Tset	Setpoint temperature of the thermostat

Deadband	Deadband in the storage tank thermostat. This parameter specifies the temperature range, with the setpoint being at the center of the range
RecircOutSe g	Outlet segment in the recirculation loop
RecircInSeg	Inlet segment in the recirculation loop
UA	Heat loss coefficient describing the jacket and fittings of the tank
Vtan	Volume of fluid stored in the tank
perQFlu	Percentage of total heat transferred to the water via the flue
perQBas	Percentage of the total heat that is transferred to the water via the base
QDotPilot	Rate of heat consumption of the pilot light
Pmains	Mains water pressure

Four component models are summarized briefly here, then described in more detail below.

1. *draHeaTra (draw heat transfer)*. This model identifies the rate of heat transfer in each individual segment of the tank caused by cold water entering the water heater during a draw. During a draw, water enters the bottom of the tank from a vertical dip tube. After exiting the dip tube, the water continues to flow downward to the bottom of the tank. The water strikes the bottom of the tank, changes direction, and begins to flow back toward the top of the tank. The design of each individual water heater affects the water flows in the tank.

2. *cirHeaTra (circulation heat transfer)*. This model identifies the heat transfer in each segment of the tank caused by a recirculation loop. Because there currently are no data for storage tanks with recirculation loops, this model has not been validated. It is available primarily to provide structure for future work.

3. *buoHeaTra (buoyant flow heat transfer)*. This model describes the heat transfer among segments caused by buoyancy-induced flows driven by heat transfer from the burner. It identifies which segments are above or below the stratification layer and allocates heat transfer from the base and the flue accordingly.

4. *uALos (UA heat loss)*. This component model identifies the heat lost to the environment through the jacket of the tank. The rate of heat transfer is determined using a heat transfer coefficient called the *UA value.*

4.1.1.1 draHeaTra

The model draHeaTra describes heat transfer caused by cold water entering the tank during a draw. The user enters the parameters perInA, perInB, perInC, timDelA, timDelB, and timDelC. Inlet water temperature, the temperature of water in the each segment, and the mass flow rate of the draw are all time-varying inputs used in draHeaTra.

The perIn and timDel values create second-order polynomial equations describing how the cold water enters the tank. The perIn values are used to generate segment-specific equations that define the percentage of cold water from the draw entering each segment of the tank. Values for timDel are used to generate a similar equation for the time until mains water begins entering each segment of the tank. A generic version of the polynomial is expressed in the following equation.

$$y[i] = A[i] * x^2 + B[i] * x + C$$

In the equation, *y* represents either the percentage of water entering a segment or the time until the mains water reaches that segment; *x* represents the mass flow rate of water; *i* is an index used to track the results in each segment; and *A*, *B*, and *C* are the parameters entered by the user (either perIn or timDel values).

The perIn and timDel parameters are specific to each individual heater and must be determined experimentally.

The perIn and timDel polynomial equations identify the mains water entering each specific segment of the tank. The volume of water entering a segment must equal the volume of water exiting that segment. Cold water entering the bottom of the tank induces upward flow from the bottom segment.

Heat flows caused by these water flows are identified using the following equation.

$$\dot{Q}_x[i] = \dot{m}_x C_p (T_{In} - T[i])$$

In this equation, $\dot{Q}$ represents the rate of energy change in a segment, *i* is an index used to track the segments, $\dot{m}$ is the mass flow rate of water, C_p is the specific heat of the heated fluid, T_{In} is the temperature of water entering the segment, and *T*[*i*] is the temperature of water in the segment.

The equation applies to both the heat flow rate caused by mains water entering a segment and the heat flow rate caused by the balancing flows. For the mains water calculations, $\dot{m}$ is the flow rate identified by the perIn and timDel polynomials, and T_{In} is the mains water temperature. For the balancing flow calculations, $\dot{m}$ is the sum of the balancing flows from all segments below the current segment, and T_{In} is the temperature of the segment below the active segment.

4.1.1.2 cirHeaTra

The component model cirHeaTra was created to simulate the heat transfer caused by water flows in a recirculation loop. The model is preliminary, and future work will add the capability to model hybrid water heaters or solar thermal systems.

The cirHeaTra model currently assumes that the rate of circulation flow entering each segment is identical. Future work will address development of a circulation flow model that allows different flows in each segment. The current form of the equation for cirHeaTra is presented in the following equation.

$$\dot{Q}_{Rec}[i] = \frac{\dot{m}_{Rec}}{nSeg} * C_p * (T_{Rec} - T[i])$$

In this equation, $\dot{Q}_{Rec}$ represents the heat transfer rate caused by recirculation flow, *i* is an index used to track each segment, $\dot{m}_{Rec}$ is the recirculation mass flow rate, *nSeg* is the number of segments in the simulation, T_{Rec} is the recirculated water temperature, and *T*[*i*] is the temperature in the current segment.

4.1.1.3 buoHeaTra

The component model buoHeaTra identifies the heat transfer rates caused by heat inputs from the pilot light and gas burner. The model relies on three time-varying inputs: the temperature of the water in each segment of the tank, the rate of heat transfer through the base, and the rate of heat transfer through the flue. The user provides values for the parameters *nSeg* and *QDotPilot*.

The calculations in buoHeaTra are performed assuming that there are two regions in the storage tank. The first is a region of well-mixed water at the bottom of the tank. The water is well mixed because heat transfer rates through the base and flue are high enough to cause buoyancy driven flows. Higher in the tank the water temperature exceeds that of the water in the well-mixed region. This point is referred to as the stratification layer. The second region describes all of the water above the stratification layer.

The model initially identifies whether a segment is above or below the stratification layer by comparing the temperatures of each segment to the temperature of the segment above it, beginning at the bottom of the tank. A segment is assumed to be below the stratification layer if the temperatures of the two segments are the same. If the temperature of the segment above the active segment is higher than the temperature of the active segment, then that segment is considered to be above the stratification layer. The model identifies the segment where the transition occurs.

The model then identifies the total heat rate below the stratification layer. It combines the heat transfer through the base and through the flue for each segment below the stratification layer. To make this calculation, the model assumes that heat transfer through the flue is constant for each segment of the water heater (Hoover 2012).

Finally, the model performs calculations to add heat to each segment in the tank. Because the region below the stratification layer is well mixed, the heat is applied evenly to all segments. Based on the assumption of constant heat transfer through the flue, the segments above the stratification layer each receive $\frac{\dot{Q}_{Flue}}{nSeg}$.

4.1.1.4 uALos

The component model uALos describes the rate of heat loss to the environment. The model divides a heat transfer coefficient (UA) by the number of segments in the tank. This calculation creates a UA term for each segment of the tank, identifying the heat transfer rate through each segment. The calculation is based on the simplifying assumption that the heat transfer through each segment is identical. Future work can develop specific UA values for each segment of the tank that will consider the additional surface area at the top.

The formula used to identify the rate of heat loss to the environment is presented in the following equation.

$$\dot{Q}_{Loss}[i] = \frac{UA}{nSeg} * (T[i] - T_{Amb})$$

In this equation, $\dot{Q}_{Loss}$ refers to the rate of heat loss, *i* is an index used to track each segment, *UA* is the heat loss coefficient for entire tank, *nSeg* is the number of segments in the tank, *T*[*i*] is the water temperature in a given segment, and T_{Amb} is the ambient temperature.

4.1.2 Validation

The validation for the non-condensing storage tank water heater model consisted of two phases: several draws at different flow rates, and a long delay with no draw. The details of the data collected can be found in Chapter 3. The first phase, consisting of several draws at different flow rates, enabled LBNL to determine how accurately the simulation model identifies stratification in the tank during draws, as well as during burns. Researchers also used these data to validate the operation of the thermostat. Results are shown Figure 30.

Figure 30: Experimental and Simulated Temperatures at Various Depths in the Storage Tank Water Heater During Several Draws

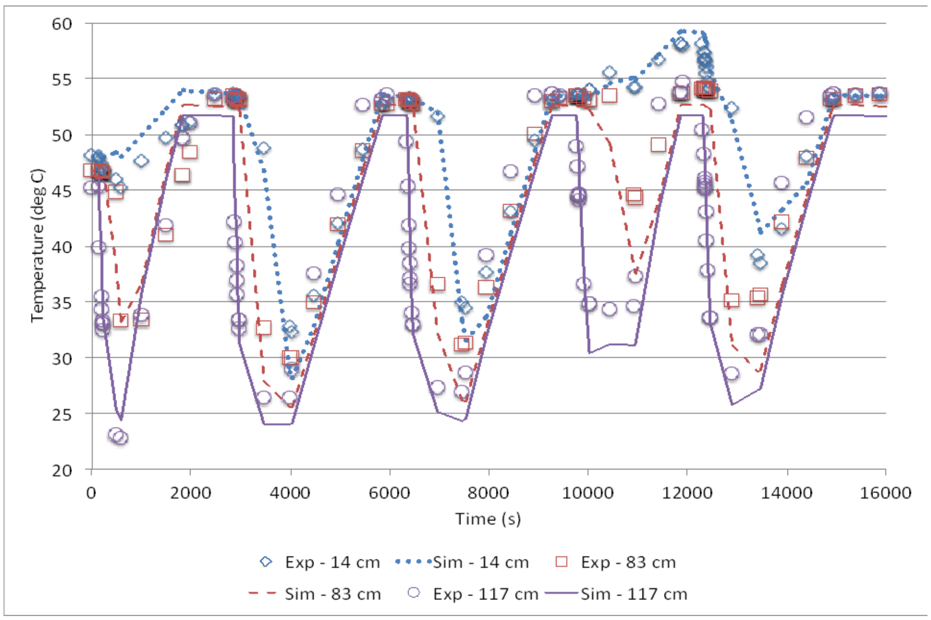
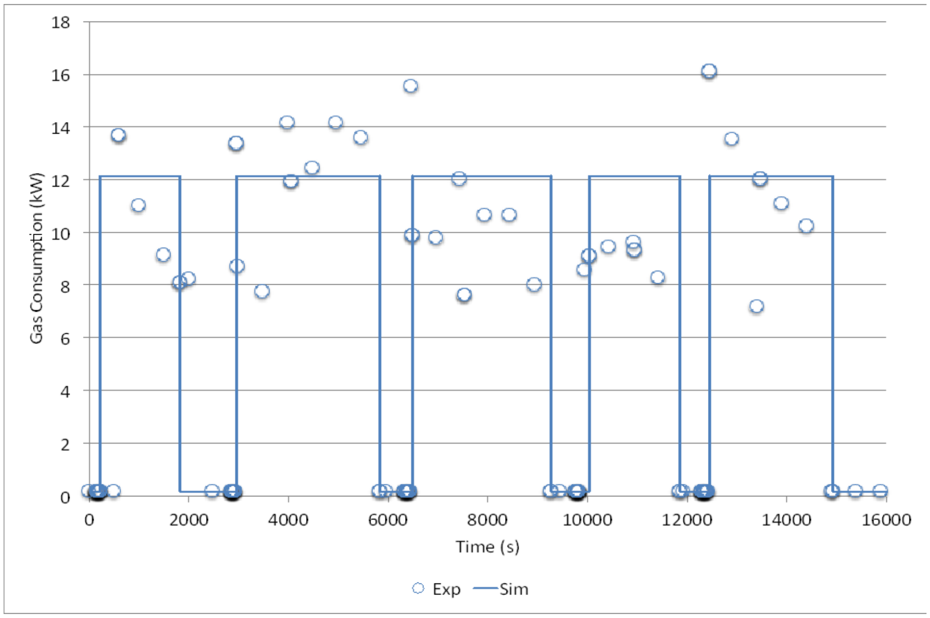


Figure 30 shows the temperature at various depths in the storage tank. Experimental data are shown using data points; simulation results are shown using lines. The same color is used to show data series that are at the same depth in the tank. The data in Figure 30 show that the simulation model accurately predicts the temperature at various depths in the tank during the repeated draws that occurred as part of the validation test.

Figure 31 shows the natural gas consumption as measured experimentally and as predicted by the simulation model.

Figure 31: Experimental and Simulated Natural Gas Consumption During the Repeated Draws of the Validation Test

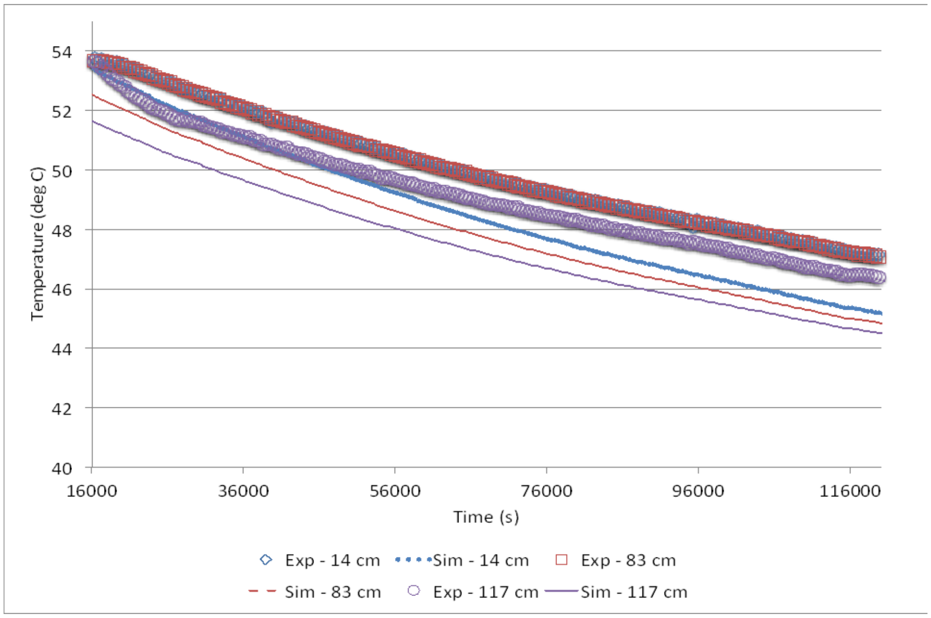


The results in Figure 31 show that the simulation model does a reasonable job of predicting the natural gas consumption during repeated draws. Some points of concern are listed below.

- The simulation model predicts that during the original draw, the burner will begin firing approximately 120 s before the experimental data show it did. This difference likely is caused by imprecision in the characterization of the thermostat.
- The natural gas consumption data were measured using a standard residential natural gas meter modified to send out 500 pulses per cubic foot of natural gas consumed. The sensors are notoriously noisy. The noise in the experimental data may require adjustment of the burner's selected heat rate. Future work will include further investigation.

The second phase of the validation test consisted of a long delay with no draws. During the delay the water was heated only by the pilot light. Because the heat loss through the jacket to the surrounding environment exceeded the heat added by the pilot light, the temperature of all segments in the tank slowly decreased. Temperature results are shown in Figure 32.

Figure 32: Experimental and Simulated Temperature at Various Heights in the Tank During a Long Delay Between Draws



The data in Figure 32 show that the simulated values are slightly lower than the experimentally measured values. Both simulated and experimental data are presented at three different depths in the storage tank. The decay length was 100,000 seconds (1.17 day). On average the simulated temperatures decreased by 7.7 °C (13.8 °F), while the experimentally measured temperatures decreased, on average, by 7.3 °C (13.1 °F). This is a 4.5-percent difference in temperature loss.

4.2 Tankless Water Heater Model

Non-condensing gas tankless water heaters currently are the fastest-growing segment of the water heating market (Parker 2011). Researchers require access to up-to-date models of non-condensing gas tankless water heaters to analyze occupant satisfaction and energy efficiency, and to propose solutions to improve whole-building energy efficiency.

Lawrence Berkeley National Laboratory has created a model of a non-condensing gas tankless water heater that is based on previous tankless water heater models (Grant 2011). It incorporated the following improvements on previous models:

- The previous model contained a standard feedback controller, but data were insufficient for tuning the controller. As described in Chapter 3, LBNL recently performed experiments to collect data specifically for tuning the PID controller. These data were used to better characterize the controller in a tested unit and match simulation results to experimental data.
- The newest version of the non-condensing tankless water heater model uses previously developed models for fluid flow rather than real variables. This change reduces the number of equations and makes the model more robust and easier for users to apply.

- To improve usability of the controller model, several component models were developed. Each component model contains all the calculations for an individual logic check. The controller in previous models was complex and difficult to understand. Previously, all calculations were performed in a single model for which documentation of internal algorithms was limited.
- We added a model to emulate the delay between a draw beginning and the warming of the heater. That model simulates the start-up process of the heater. Typical non-condensing gas tankless water heaters have two start-up control sequences: a rapid start-up and a slow start-up. The time between draws determines whether the heater uses the rapid or slow start-up. The new model enables users to specify the time required for both the rapid and slow start-up sequences, as well as the delay time before the heater utilizes the slow sequence.
- A model has been added to the controller characterizing the electric energy consumption of the heater.
- Preliminary work has been done to include sensor delay in the controller, as well as to allow efficiency to be a time-varying input. A model has been created to identify efficiency as a function of entering water temperature, which will be necessary when developing models for condensing water heaters.

The non-condensing gas tankless water heater model contains two component models: one for a heat exchanger, and one for a controller. The heat exchanger model determines the heat absorbed, heat transferred to the water, and heat lost to the environment. The model also accounts for inefficiency losses. The controller model simulates the logic in the controller of a typical non-condensing gas tankless water heater. Several experimentally derived parameters are entered in the model to describe the heat exchanger and controller for a specific water heater. The parameters are described briefly in Table 7.

Table 7: Parameters for Non-Condensing Tankless Water Heater Model

Medium	The fluid being heated. By default this references a model for water in the LBNL Buildings Library.
Capacitance	Thermal capacitance of the heat exchanger
SteadyStateEff	The steady-state efficiency of the unit
Q_flow_rated	Rated maximum heat rate
Q_flow_min	Rated minimum heat rate
m_flow_min	Minimum fluid flow rate of the heater
SpecificHeat	Specific heat of the heated fluid
PID_P	P value in the PID feedback controller
PID_I	I value in the PID feedback controller
PID_D	D value in the PID feedback controller
nNode	Number of nodes in the heat exchanger
EffPilot	Efficiency of the pilot light
Q_flow_pilot	Rate of heat consumption of the pilot light
t_short	Duration of the rapid start-up sequence
t_long	Duration of the slow start-up sequence
t_switch	Time between draws before slow start-up sequence is used
QAct	Electricity consumption when the fan is active
QIdl	Electricity consumption when the fan is idle

4.2.1 Heat Exchanger

The heat exchanger model uses inputs from the controller model and parameters specified by the user to perform the following calculations:

- Identify the rate of heat gain from the burner in each node. This calculation multiples the control constant (which is an input from the controller model), the steady-state efficiency, and the maximum heat rate of the unit. The result is the total heat rate of the unit. The heat rate then is split evenly among all nodes by dividing by nNode.

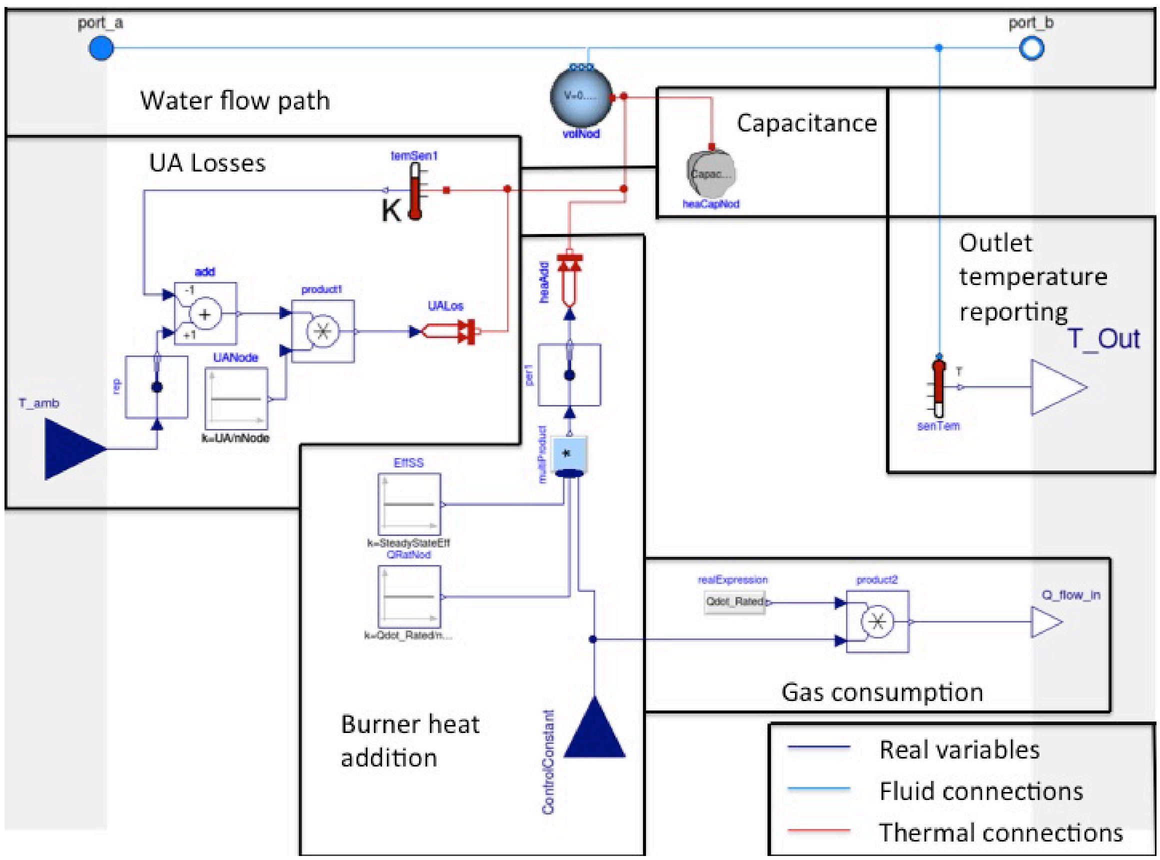
- Identify the heat loss to the surrounding environment in each node. To obtain the UA term for each node, divide the total UA term by nNode. Multiplying the UA term with the temperature difference between the segment and the ambient temperature will yield the heat loss from each segment.

- Identify the temperature of each segment in the heat exchanger. Because it is not feasible to collect measurements for comparing the temperature of water in the heat exchanger to the heat exchanger itself, the water and heat exchanger are modeled using a lumped capacitance.

- Identify the rate of heat consumption. This rate is found by multiplying the control signal (from the controller) with the rated maximum heat rate.
- Report the outlet water temperature. This calculation provides the user with the temperature of water leaving the heat exchanger and sends the necessary signal to the controller model.

Figure 33 shows a schematic of the model for the non-condensing heat exchanger used in the non-condensing tankless water heater model.

Figure 33: Heat Exchanger Model Used in the Non-Condensing Tankless Water Heater Model



In Figure 33, black boxes differentiate sections in the model, and black text describes the function of each section. The water flow path, which is at the top of the schematic, contains three models. From left to right they are: a fluid inlet port, a mixing volume for the fluid, and a fluid outlet port. The fluid inlet and outlet ports allow connection to other models along the fluid path. The mixing volume model represents the water in each node of the heat exchanger. The light blue lines connecting the three components represent a fluid flow path.

A section containing a model of thermal capacitance is shown just down and to the right of the mixing volume model. The thermal capacitance model was developed assuming that the water in a node of the heat exchanger is at the same temperature as the heat exchanger itself. The thermal capacitance model represents the combined capacitance of the water and the heat exchanger. The red line connecting the capacitance to the volume allows heat to transfer between the two models while maintaining them at the same temperature.

The section on the left labeled *UA Losses* identifies the heat loss rate from each node of the heat exchanger to ambient air. It reads the temperature of each node, then compares that to the ambient temperature. The model then multiplies the temperature difference by the UA term for each segment and informs the rest of the model of the heat loss rate.

The section that is directly below the mixing volume, labeled *Burner heat addition*, identifies the heat rate added to each segment of the heat exchanger by the burner. This section of the model reads the control signal coming from the controller, multiplies that value by the rated heat rate, multiples by the steady-state efficiency, and divides by the number of segments to identify the heat added to each segment. It then prescribes that heat addition to each node of the heat exchanger.

The final two sections, *Outlet temperature reporting* and *Gas consumption*, report model results and pass those values out of the heat exchanger model.

4.2.2 Controller

The second component model used in the non-condensing gas tankless water heater model is for the controller. The controller model identifies whether the burner should fire, at what percentage of capacity the burner should fire, and the electricity consumption of the heater. Figure 34 shows a schematic of the controller model.

Figure 34: The Controller Model Used in the Non-Condensing Tankless Water Heater Model

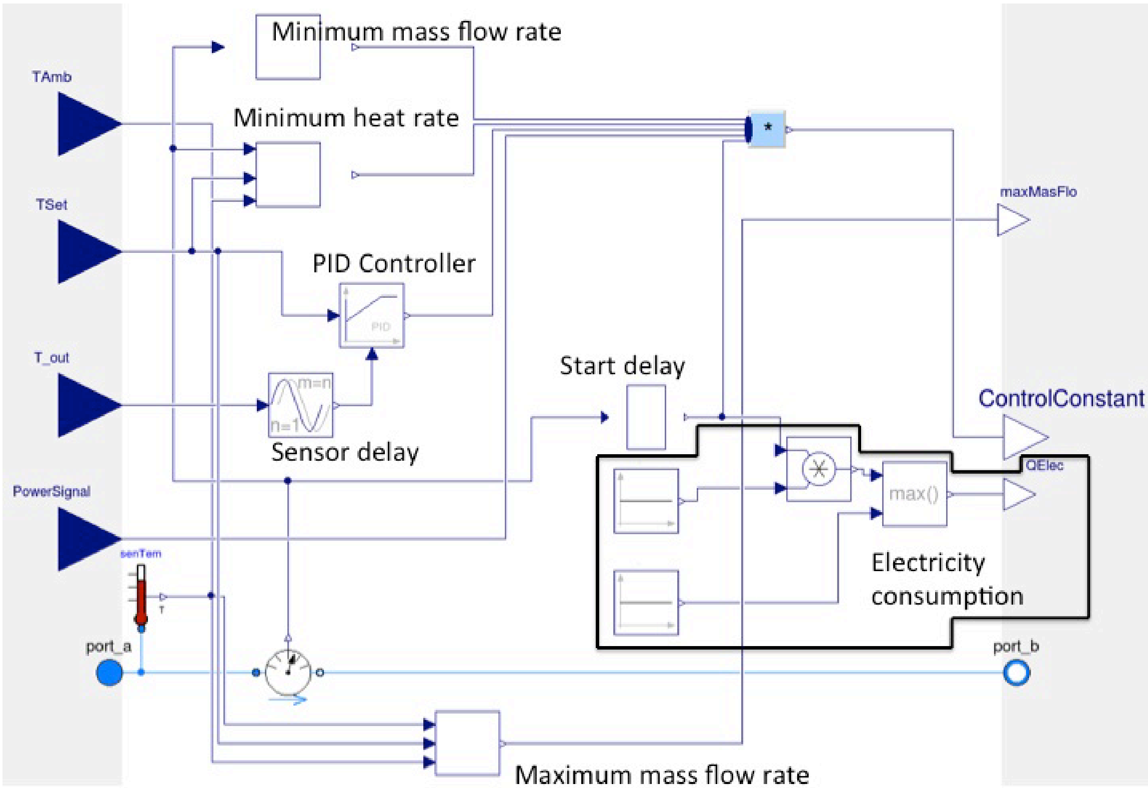


Figure 34 shows several improvements over the previous version of the controller model for a non-condensing tankless water heater. The previous model contained all the programming for each logic check at the top level. In the new model, component models contain much of the programming, making the model easier for users to understand and modify. Those component models are described below.

The *Minimum mass flow rate component model* compares the current fluid flow rate to the minimum flow rate specified for the heater. If the water flow rate is less than the minimum flow rate, this model instructs the controller that the burner should not fire.

The *Minimum heat rate component model* identifies the required heat rate to bring the water flow to the setpoint, compares that water flow to the minimum heat rate specified for the heater, and instructs the controller on whether the burner should fire. The burner will not fire in situations where the required heat rate is less than the minimum heat rate. This model replaced several calculations within the controller model. The addition of this model both makes the current model easier to understand, and makes it easier for users to model future controllers with minimum heat rate checks.

The *PID controller component model* identifies what percentage of the maximum heat transfer rate the burner should consume under time-varying circumstances. It is a model for a common feedback controller used in many control processes.

The *Sensor delay component model* was included to provide future capabilities, and is not currently in use. It delays the temperature signal coming from the outlet temperature sensor, capturing the effects of sensor lag. Future work will include re-tuning the PID controller to account for the sensor lag.

The *Maximum mass flow rate component model* identifies the maximum water flow rate the system can handle under current conditions. It uses the inlet temperature, rated maximum heat of the heater, and steady-state efficiency of the heater to identify how much water can be brought to set temperature. It then outputs that flow rate as information for other models to use. Some tankless water heaters incorporate flow restriction devices. The flow restriction devices can be modeled by adding a pressure drop in the system that reduces the system flow to match the maximum flow rate.

The *Start delay component model* accounts for the start-up process in the heater. The tested heater included both a rapid and a slow start-up process. The heater chooses which process to use depending on the delay between the active draw and the previous draw. The start delay model allows the user to specify a duration for the rapid sequence, a duration for the slow sequence, and a time delay between draws that selects the appropriate sequence. The model then delays burner firing for the correct period of time at the start of a draw.

The *Electricity consumption component model* identifies the rate of electricity consumption at any time. The user can specify values for fan electrical consumption when the heater is active and when it is idle. The model uses the active value when there is a draw, and the idle value when there is not.

4.2.3 Characterization

Lawrence Berkeley National Laboratory researchers characterized the water heater using data collected from the experiments described in Chapter 3, and identified parameters to describe the tested unit for the non-condensing tankless water heater model. The characterization process involved three steps: identify the steady-state efficiency and duration of start-up sequences for the heater through spreadsheet calculations; use an optimization script in the Modelica Design package calibration function (Elmqvist et al. 2005) to vary the parameters to match experimental data, and refine the UA value to match experimental data.

4.2.3.1 Steady-State Efficiency

Lawrence Berkeley National Laboratory calculated the steady-state efficiency for five different draws. The data set included draws at three different flow rates and three different set temperatures. This variation enabled researchers to determine whether steady-state efficiency is a function of water flow rate or setpoint. Table 8 presents the data collected regarding steady-state efficiency.

Table 8: Steady-State Efficiency as a Function of Water Flow Rate and Temperature Setpoint

m_flow_nominal (kg/s)	T_Out (°C) / (°F)	Efficiency
0.08	37.78 / 100.00	0.81
0.13	37.82 / 100.08	0.80
0.19	37.84 / 100.11	0.86
0.13	43.51 / 110.32	0.85
0.13	48.08 / 118.54	0.83

The data in Table 8 show no correlation between flow rate or setpoint and efficiency. As a result, researchers used the average of the steady-state efficiency values in the characterization. The average recorded steady-state efficiency is 82 percent, which exactly matched the manufacturer's published percentage (Rheem, no date).

4.2.3.2 Duration of Start-Up Sequences

Lawrence Berkeley National Laboratory also analyzed data to identify the duration of the start-up sequences used in the tested heater. Figure 35 shows the duration of the start-up sequence compared to the delay between draws.

Figure 35: Duration of Start-Up Sequence Given Various Delays Between Draws in a Tested Rheem84-DVLN

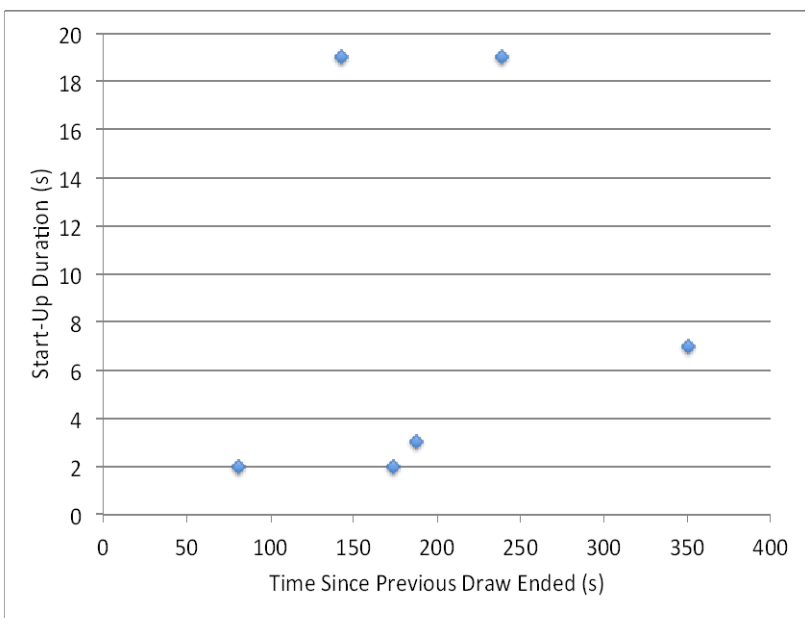


Figure 35 does not show the two clear start-up sequences that were expected. Instead it shows five durations of start-up protocol, without a clear transition from one to the next. The data in Figure 35 indicate that further work is needed to better understand the start-up procedure.

Lawrence Berkeley National Laboratory used the values in Table 9 to simulate the results shown in Figure 35.

Table 9: Start-Up Duration Parameters Used in Characterization of Rheem 84-DVLN

Parameter Name	Time (s)
t_short	2
t_delay	13
t_switch	200

4.2.3.4 Vary Parameters to Match Experimental Data

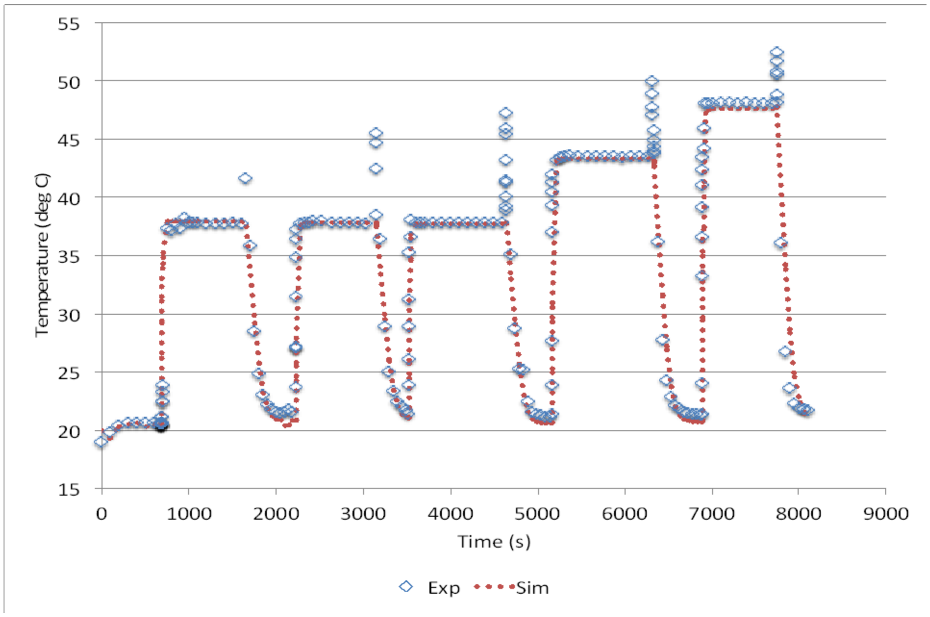
The second phase of the characterization focused on identifying the P, I, and D constants in the PID controller and the capacitance of the heat exchanger. A calibration script adjusted the parameters until the simulation results closely matched experimental data described in Chapter 3. Table 10 shows the parameters determined by the calibration script.

Table 10: Parameters Describing the Rheem 84-DVLN Heater During Characterization

Capacitance (J/C)	7,505
PID_P	0.85
PID_I (s)	12,086
PID_D (s)	9.55

Figures 36 and 37 present data that compares the simulation model and experimental results from the characterization process. The simulation was performed using the parameters in Tables 9 and 10, as well as the 82 percent steady-state efficiency.

Figure 36: Outlet Temperature Data Comparing Simulation (Sim) and Experimental (Exp) Results in the Characterization Protocol



The data in Figure 36 show that the simulated outlet temperature closely matches the experimental measurements during the draw. During both the warm-up and steady-state periods, the simulation results closely agree with the experimental data. The simulation results do not agree with the experimental data at the end of a draw, however. When a draw stops, a delay in the sensors prevents the heater from correctly responding for a short period (on the order of 1 to 2 seconds). During this period the heater continues adding heat to the heat exchanger, but without water flow the temperature in the heat exchanger increases dramatically. This effect currently is not captured in the simulation model. A sensor delay component model, which can capture this effect, will be included in future work.

Figure 37: Gas Consumption Data Comparing Simulation (Sim) and Experimental (Exp) Results During the Characterization Protocol

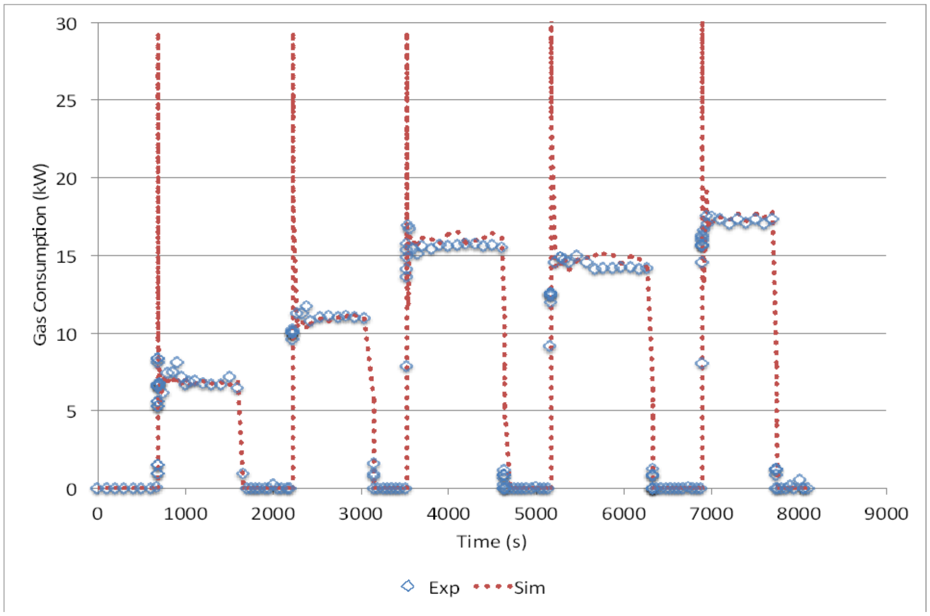


Figure 37 shows the data for natural gas consumption from both the simulation model and experimental data. The data show that the new PID constants capture transient effects at the start of a draw, an improvement over previous models. However, the simulation model still predicts a much larger draw of natural gas at the very start of a draw than indicated by the experimental data. Future efforts will include re-tuning the PID controller after implementing the sensor delay and further improving the P, I, and D constants.

4.2.3.5 Adjust UA Value

In the third step of the characterization, LBNL adjusted the UA value to match experimental data described in Chapter 3. Researchers adjusted the UA value until the temperature decay profile predicted by the simulation model matched the profile identified in the experiments. Because the capacitance of the heater was identified in the second phase of characterization, the initial temperature of the water determined the accuracy of the temperature decay profile. Table 11 shows the initial temperatures from both the experiment and simulation after several different delays.

Table 11: Temperature in Heat Exchanger After Delays Between Draws, as Measured In Experiments and Predicted in Simulations

Length of Delay (min)	T_Exp [C]	T_Sim [C]	dT [C]
6.20	42.01	41.05	-0.96
10.12	41.64	41.05	-0.59
29.87	36.86	37.31	0.45
77.18	30.56	30.28	-0.28

The data in Table 11 show that (1) the simulation model accurately predicts heat lost during delays between draws, and (2) the model predictions become more accurate with longer delays. As previously discussed, the simulation model, without a sensor delay model, does not capture the heat addition at the end of a draw. Consequently, the simulation model starts from a lower temperature than do the experiments. This lower temperature, in turn, results in a simulated UA value that is higher than the actual UA value in order to increase heat loss rates and reduce the outlet temperature in the simulation to match that of the draw. The observed error is larger for shorter delays than in larger delays.

4.2.4 Validation

Lawrence Berkeley National Laboratory validated the model of the non-condensing tankless water heater by comparing it to the experimental data described in Chapter 3. The validation consisted of two scenarios. The first involved several draws at different flow rates, with no pause between draws; instead, a sudden change in flow rate was introduced during the same draw. The second scenario involved draws separated by long periods.

4.2.4.1 Scenario 1: Several Draws at Different Flow Rates

By comparing simulation results to the experimental data, researchers were able to check the behavior of the simulation models during steady-state draws and during sudden changes in flow rate. Figure 38 shows the data for outlet temperature during the first phase, and Figure 39 shows the data for natural gas consumption.

Figure 38: Experimental (Exp) and Simulated (Sim) Outlet Temperature During a Constant Draw at Changing Flow Rates

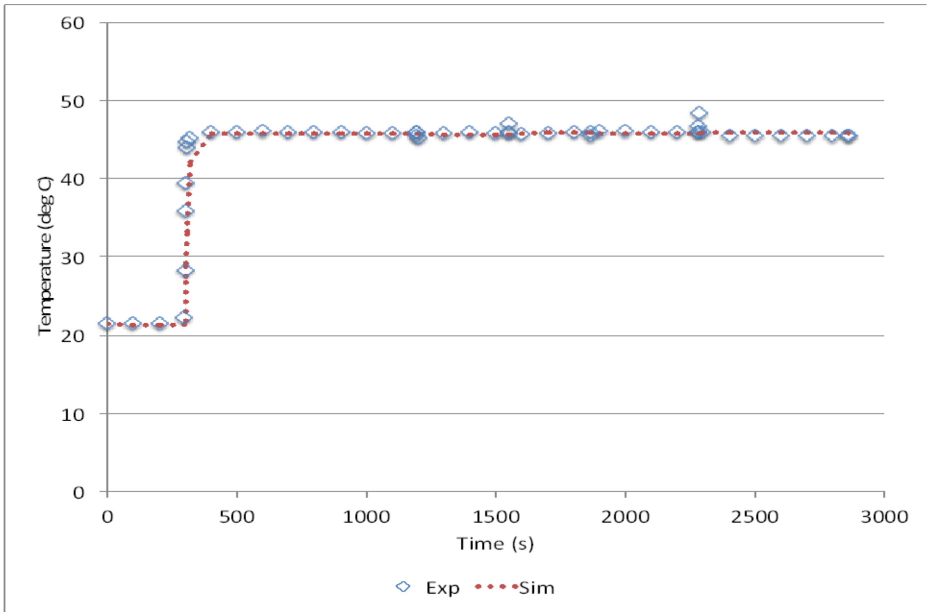
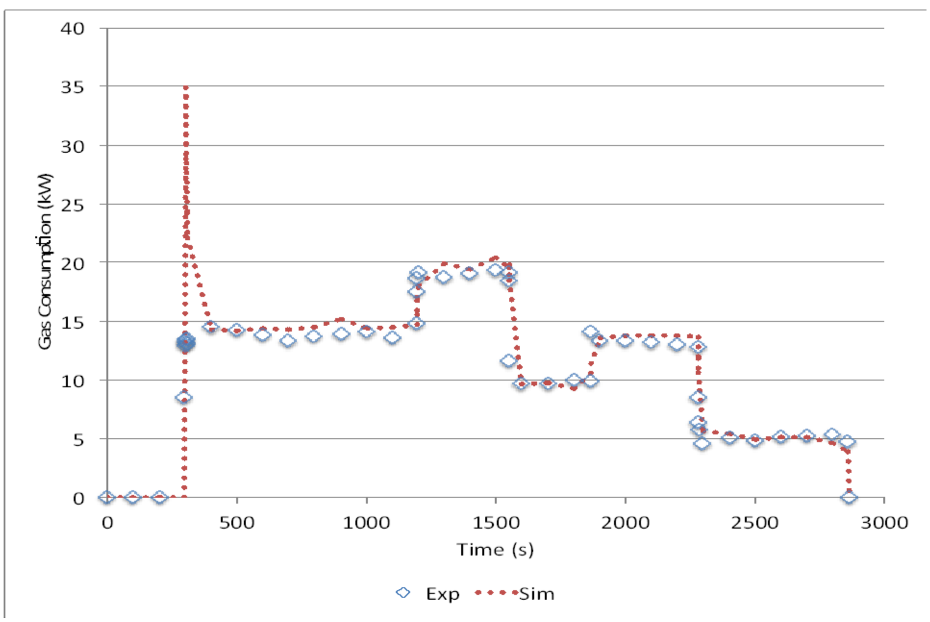


Figure 39: Experimental (Exp) and Simulation (Sim) Natural Gas Consumption Results During a Constant Draw at Changing Flow Rates



The results in Figures 38 and 39 are consistent with the results obtained from the characterization phase. In most cases, the simulation model accurately predicted outlet temperature. The model did not accurately capture the changing temperature when the flow rate changed; however, the temperature change in the experimental results was on the order of

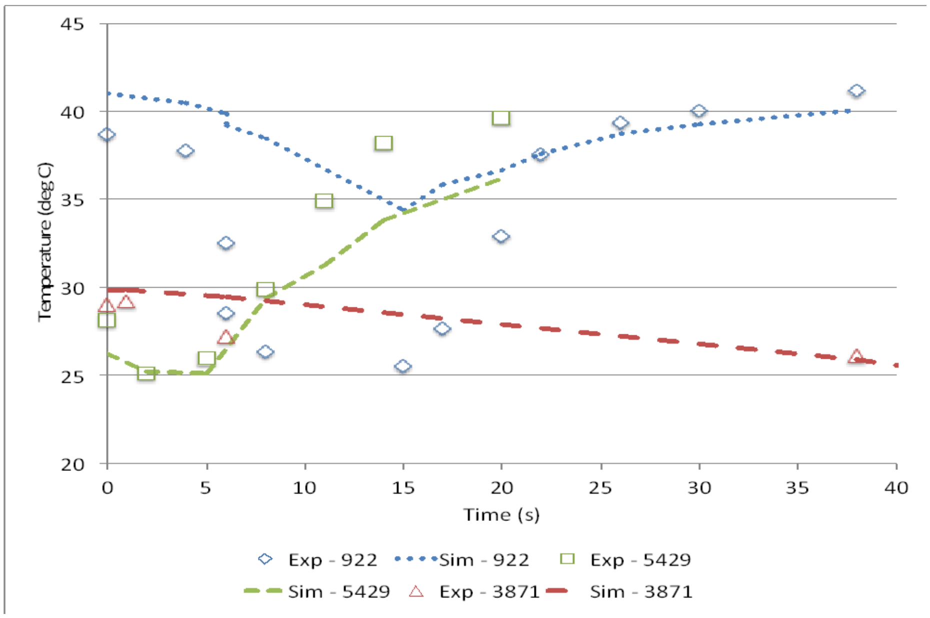
1 °C (1.8 °F). Future work implementing the model component for the sensor delay will enable more accurate depiction of transient effects.

The gas consumption results in Figure 39 are consistent with results obtained from the characterization phase. The simulation model overestimated natural gas consumption at the start of the draw but accurately predicted the gas consumption during the steady-state portion of the draw. The simulation model accurately predicted the gas consumption during the rapid changes in flow rate.

4.2.4.2 Scenario 2: Draws Separated by Long Decay Periods

The second scenario compared the simulation results to experimental data obtained during draws having long decay periods between them. Draws were performed after delays of 922 seconds (15.4 min) and 5,429 seconds (90.5 min). The data also included a very slow draw, whereby water was pushed out of the heat exchanger without the burner firing, after 3,871 seconds (64.5 min). Figure 40 shows temperature results during the long decay part of the validation test.

Figure 40: Experimental and Simulation Temperature Results During the Long Decay Part of the Validation Process



In Figure 40 the times of all three draws are normalized to the start of the water draw event (water began flowing at t = 0 in all three instances). The results show that the simulation model accurately predicted the outlet temperature profile during the draws after 5,429 seconds (90.5 min) and 3,871 seconds (64.5 min). The model underestimated the temperature decrease in the 922 seconds (15.4 min) decay, however.

4.2.5 Future Work

Some capabilities could be improved or added to the models to expand their scope and usefulness. Some potential modifications to the models and the kinds of studies they would allow are described briefly below.

- Both models contain the basic code necessary to develop models for condensing water heaters. If experimental data were available to verify the models' treatment of changing efficiencies and characterizing the heaters, researchers could develop models of condensing water heaters. Creating models of condensing water heaters would allow all of the studies performed to date to include higher-efficiency condensing water heaters.
- The storage tank water heater model cirHeaTra could be improved to identify the mixing in the tank caused by water flow. Doing so would allow simulations of solar thermal systems or hybrid water heaters. Both technologies are becoming more prevalent in the market, and accurate simulation tools would allow researchers to analyze novel designs.
- The model for the tankless water heater currently does not model sensor delay. It also assumes a two-stage response delay model, when testing implies that the time delay controls may be more complex. Including sensor delay, improving the start delay model and refining the assumption that UA is evenly distributed across the nodes would improve simulation results both at the start and the end of draws. The PID constants should also be re-tuned, as the model inaccurately predicts a large spike in gas consumption at the start of each draw. Improving those aspects would enable more accurate simulation results for the start and end of draws. Improved results at the start and end of draws would enable researchers to better study occupant satisfaction (as defined by the time necessary for the water to reach steady-state at the setpoint) and better predict the heat loss between draws.

Lawrence Berkeley National Laboratory's water heating simulation models enable researchers to study questions that have previously been unanswerable. Some of the studies that are now possible are described briefly below.

- Because the water heating section of the current Title 24 is not based on simulation studies, there is no way to identify its effectiveness. Simulation studies, using LBNL's new models, can support significant increases in water heating energy efficiency throughout California.
- The in situ efficiency of tankless water heaters currently is a major discussion topic in the hot water community. Tankless water heaters are not as efficient as predicted by the U.S. DOE energy factor test; however, no significant study has been performed to identify the effect of draw profile on efficiency. Performing such a study would enable researchers and code developers to better identify an effective efficiency derate factor and allow installers to better determine whether to install a storage tank or tankless water heater.

- Base models are available to study water heater redesign. By changing parameters or quickly modifying control logic, simulation can identify how much energy could be saved by adding insulation to water heaters, decreasing capacitance of tankless water heaters, or varying the control strategy. Such studies can inform manufacturing decisions or be used to require higher-efficiency models, and thus drive design changes.

Chapter 5: Simulation Model for Hot Water Distribution Systems

This chapter presents the results of modeling efforts addressing the hot water distribution system. These simulation tools eventually will be included in the LBNL Buildings Library for use in whole-building energy simulations (Wetter 2011). Lawrence Berkeley National Laboratory has created models in the Modelica language that can be used to perform simulations of a hot water distribution system (HWDS). The HWDS package utilizes a drag-and drop methodology. Several component models were created to describe parts of the system. Users are able to rapidly construct their own models by dragging and dropping the necessary components to describe their situation. The models have varying levels of complexity, allowing users to choose between more detailed results or faster simulation times. Creating models for the hot water distribution system in Modelica, using the Dymola development studio, allows easy connections with the water heater models (Tiller 2001; Dynasim AB 2009)

Figures 41 and 42 show examples of systems that can be simulated using the HWDS model.

Figure 41: Example Model Simulating a Residential Hot Water System Including a Hot Distribution System

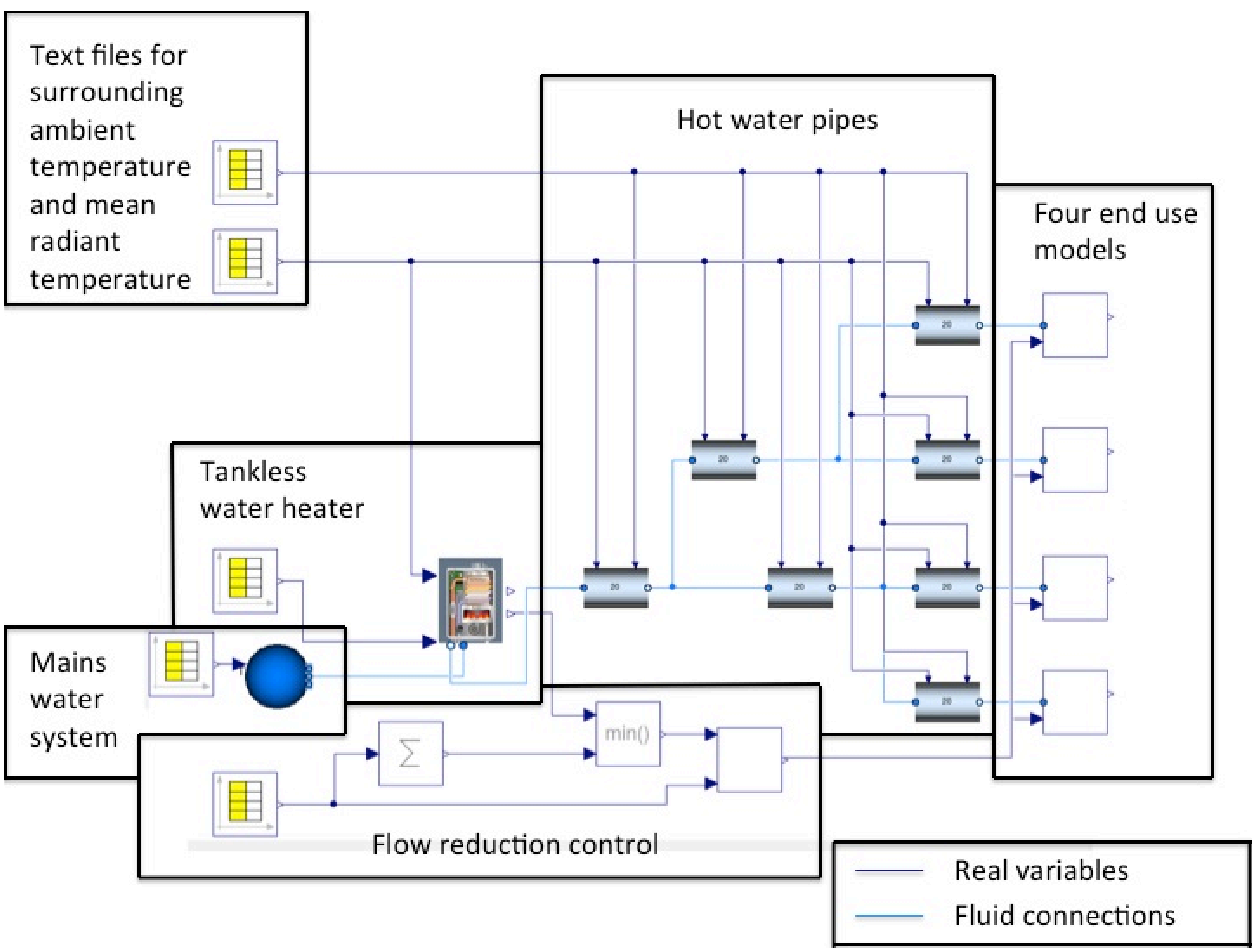


Figure 41 depicts a simulation where a non-condensing tankless water heater provides hot water for a house. A user could upon examining the hot water distribution system, decide to simplify the model by neglecting cold water flows. Component models for the HWDS simulate the water in the pipes between the water heater and end-uses. Component models, which will be described further below, include the PipeLumpedCap model, used to model a trunk-and-branch hot water distribution system. Four instances of the end-use model can simulate four separate hot water fixtures. In situations where the flow rate exceeds the capacity of the tankless water heater, the flow rate is reduced using the FlowReduction model, as some tankless water heaters do in such situations.

Figure 42: Example Model Simulating a Residential Hot Water System Including Both Hot and Cold Water Piping Systems

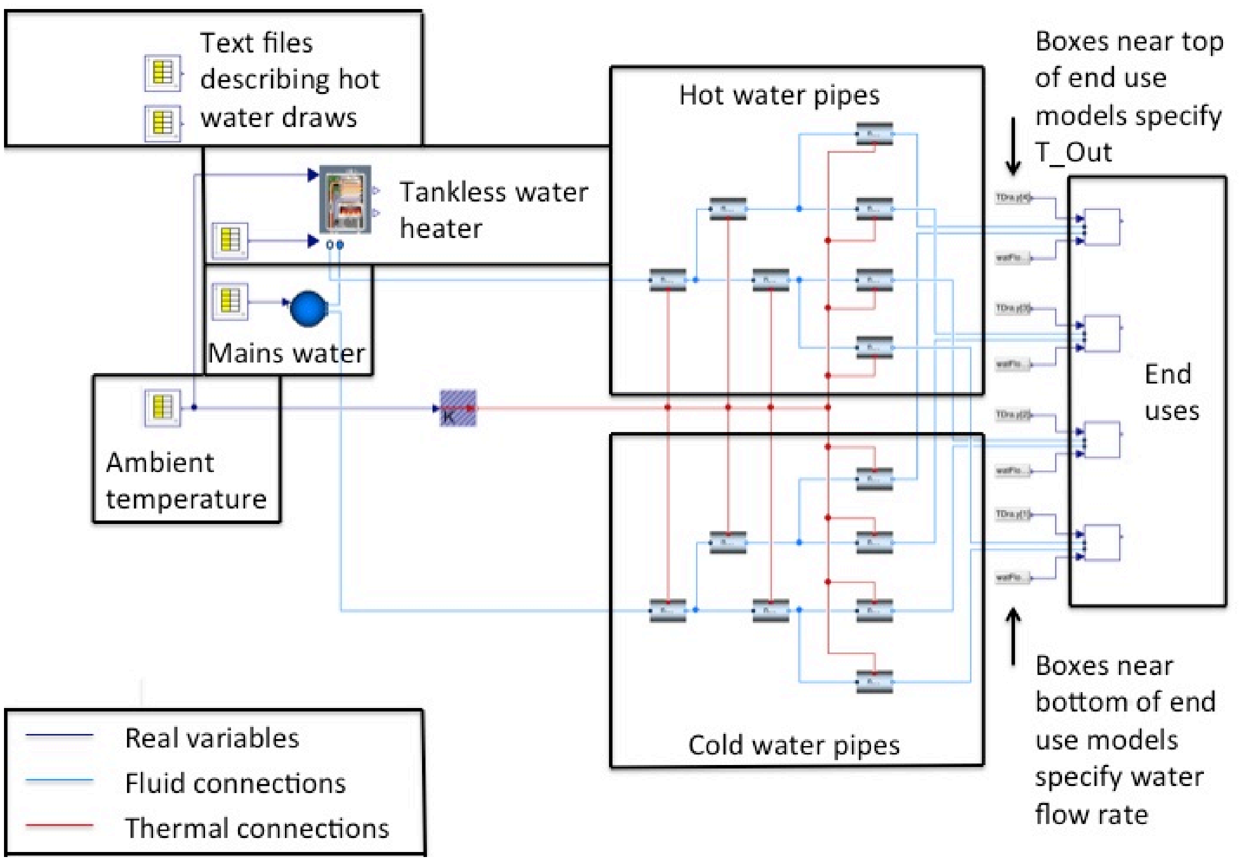


Figure 42 depicts a model that includes both hot and cold water distribution systems. As in the Figure 41, a tankless water heater serves the house. There are four primary differences between Figures 41 and 42.

- In Figure 41 the water from the mains system passes through the tankless water heater, into the hot water pipes, and out to the end uses. In Figure 42 that path is maintained while a second water path is added from the mains system through cold water pipes and out to the end uses. This additional piping path is used to model the cold water distribution system in the house.
- The PipeLumpedCap model was used for Figure 41. The addition of cold water pipes gives the system in Figure 42 twice as many pipes as the system in Figure 41. In addition, mixing valves are added to each end-use model. Because the simulation in Figure 42 models a more complex system, it was necessary to use a simpler pipe model to speed up simulation time. In Figure 42 the BasePipe model is used for both hot and cold water pipes. The mixing valves change the flow rate of hot and cold water to the end use as necessary to maintain the set temperature. In some situations the valves will stop flow in one path or reduce hot water flow until it falls below the minimum allowed by a tankless water heater. Those changes introduce state events, instances where the status of a true/false statement changes, which significantly slow down the simulation. The additional pipes add several new non-linear equations, making it difficult for the simulation to re-initialize at every state event. The non-linear equations arise from the pressure dependency of the system. Pressure loss calculations across the pipe models are inherently non-linear. Additional non-linear equations arise because the flow in one pipe depends on the flow through another. This combination can slow the simulation dramatically. BasePipe is described further below.
- The end-use model represented in Figure 42 is EndUseTwoBranch, which was designed to emulate a fixture in a two-branch system where the user specifies an outlet flow rate and temperature. The model then identifies how much water flow comes from each pipe. With EndUse (Figure 41), the user specifies the hot water flow rate, and the model neglects the cold water part of the distribution system. A detailed description of EndUseTwoBranch is provided below.
- The model in Figure 42 does not include the FlowReduction model.

5.1 Pipe Models

Research into residential hot water systems is beginning to focus on the hot water distribution system (HWDS). Current discussions are focused on the amount of energy that is lost in the distribution system and how occupant behavior affects this energy waste (Kosar et al. 2013). The fastest and cheapest way to accurately answer such questions is through use of validated simulation models of HWDS. Simulation models of generic pipes have been created that can be used to analyze heat stored within pipes and heat lost to ambient conditions. The pipe models can be used to construct distribution systems and study such topics as the temperature of water inside the pipes, the amount of energy lost to the environment, and the temperature of water supplied to an end use.

Several pipe models offer different advantages and disadvantages, depending on the user's needs. All the pipe models are based on the Pipe model in the LBNL Buildings Library. The models are described below.

1. *BasePipe*. This model is the same pipe model as in the Buildings Library. This model is able to model pressure drop across a pipe, the volume of water contained within the pipe, and insulation surrounding the water. The water inside the pipe is divided into multiple segments, and it is assumed that the water within each segment is perfectly mixed. The primary strength of this model, compared to the other pipe models, is that it is a simpler model having fewer state events and non-linear equations. State events and non-linear equations dramatically increase computation time needed by the ordinary differential equation (ODE) solvers. This simplified model results in faster simulation times and can be used to simulate highly complex systems. Weaknesses of this model include the assumption that water is perfectly mixed, the fact that there is no model for the thermal mass of the pipe surrounding the water, and the fact that the insulation is modeled as a total resistance, removing the capability to compare convective and radiative heat losses.

2. *PipeR*. The PipeR model upgrades the BasePipe model by adding (a) a capacitance model to describe the thermal mass of the pipe itself and (b) a mixing coefficient parameter. The PipeR model, which relates to the water and insulation, uses the geometrical and material properties of the pipe to model the thermal mass of the pipe wall. The pipe wall currently is modeled as a lumped mass instead of in series with the thermal mass of the water. Future research will address the construction of the pipe wall to place it in series with the water. The second change, the addition of a mixing parameter, has a value between zero and one specified by the user to dictate how effectively the water in each segment of the pipe is mixed. Zero forces plug flow in the simulation; one forces ideally mixed flow; and a value in between forces a flow having a user-defined amount of mixing. This model is more realistic than BasePipe in that it more accurately models thermal mass in the system and more accurately represents the temperature rise when hot water reaches the outlet. It is weaker than BasePipe in that the implementation of the mixing coefficient adds state events to the model, resulting in significantly longer simulation times.

3. *PipeLumpedCap*. The PipeLumpedCap model is a modified version of the PipeR model. The PipeR model handles the heat transfer between the pipe and the environment via a description of the insulation to match a certain R-value. The R-value takes into account conduction through the insulation as well as conduction and convection on the outside of the insulation. The PipeLumpedCap model breaks the heat transfer into three separate components, conductive, convective, and radiative. The user is expected to enter parameters that describe the insulation itself (conductivity and thickness). The model then separately calculates convection and radiation between the insulation and the surroundings. The theoretical advantage of this model is that the user can identify the difference between convective and radiative losses; however, sufficient data for validating this model are not available. To date it has been validated on the basis of total

heat loss, but there are no data to make sure that the convection and radiation heat loss rates are accurate. Radiation calculations are based on T^4, meaning that this model implicitly uses more non-linear equations than PipeR and requires significantly longer simulation times.

4. *PipeLumpedCapNoInsul*. This model is similar to PipeLumpedCap except that the model for the insulation has been removed. Using the PipeLumpedCap model to perform simulations for bare pipe will produce errors, so the PipeLumpedCapNoInsul model should be used when there is no insulation. PipeLumpedCapNoInsul maintains key features of PipeLumpedCap, including the handling of thermal mass, the mixing coefficient, and the split between convection and radiation heat transfer.

5.1.1 Validation of Pipe Model

Lawrence Berkeley National Laboratory compared results from the PipeLumpedCap model against experimental data collected in a test lab (Hiller 2011a, b). Validation comparisons were performed at two water flow rates and with two insulation levels on the pipes. The combinations of test conditions used in the validation are as follows:

1. 0.063 L/s (1 gal/min) water flow rate; no insulation; 2.22 cm (7/8 in, 3/4 in nominal) diameter copper pipe
2. 0.063 L/s (1 gal/min) water flow rate; 0.8277 m^2-K/W (4.7 h-ft^2-F/Btu) insulation; 2.22 cm (7/8 in, 3/4 in nominal) diameter copper pipe
3. 0.126 L/s (1 gal/min) water flow rate; 0.8277 m^2-K/W (4.7 h-ft^2-F/Btu) insulation; 2.22 cm (7/8 in, 3/4 in nominal) diameter copper pipe

The three tests, although not a thorough validation, have enough changing parameters to provide some confidence that the model is simulating the physical processes correctly.

Figure 43 shows a sample data set from the validation process. The displayed data are from an experiment performed with a 0.063 L/s (1 gal/min) water flow rate and 0.8277 m^2-K/W (4.7 h-ft^2-F/Btu) insulation on a 2.22 cm (7/8 in, 3/4 in nominal) diameter copper pipe. Parameters used to describe the pipe are provided in Table 12.

Table 12: Parameters in Validation Simulation

Parameter	Value	Unit	Significance
nSeg	46		Number of simulated volume segments in the pipe
MixCoef	0.6		Defines the extent of mixing in the pipe. 1 = complete; 0 = plug flow
ePipe	0.95		Emissivity of the pipe or insulation (insulation if insulated, pipe if no insulation)
diameter_i	2.06 (0.811)	cm (in)	Inner diameter of the pipe
diameter_o	2.22 (0.875)	cm (in)	Outer diameter of the pipe
til	0	deg	Tilt of the pipe (0 deg = horizontal)
length	27.88 (91.45)	m (ft)	Length of the pipe
c_p_pip	390 (0.092)	J/(kg-K) (Btu/(lb-F))	Specific heat of the pipe material
rho_pip	8,960 559	kg/m^3 lb/ft^3	Density of the pipe material
thicknessIns	1.2 (0.47)	cm (in)	Thickness of the insulation
lambdaIns	0.05 (0.0289)	W/(m-K) $Btu/(hr\text{-}ft^2\text{-}F)$	Conductivity of the insulation

The mixing coefficient (MixCoef) listed in Table 12 was identified as part of the validation process. After simulations were performed using several different mixing coefficients, the results were compared against experimental data (Hiller 2011a,b). The mixing coefficient affects how much incoming hot water mixes with the cold water already in the pipe. Graphical evidence of this effect is shown in Figure 43.

Figure 43: Comparison Between Experimental and Simulated Water Temperature Leaving a Pipe Given Various Mixing Coefficients

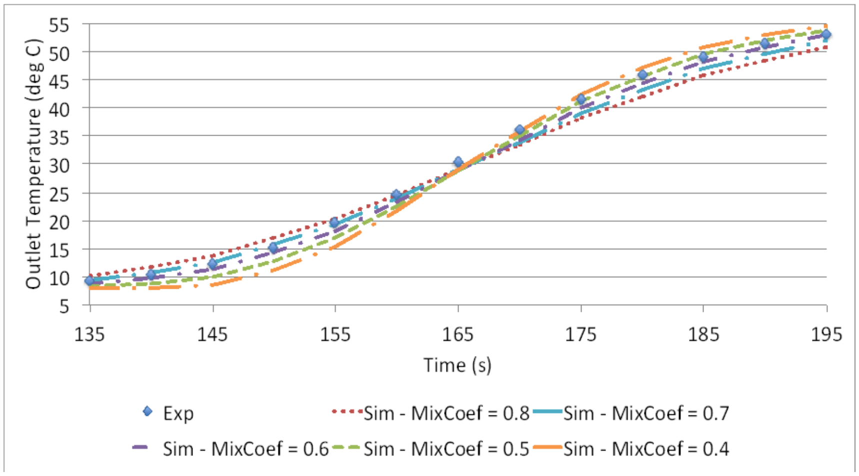


Figure 43 shows part of a data set from the test lab and the results of attempts to match simulation results to the data. At the start of the test, the water temperature in the pipes was approximately 8 °C (46.4 °F). A draw of 0.063 L/s (1 gal/min) and 58 °C (136.4 °F) was initiated at t = 0 seconds. The period shown in Figure 43 begins when the water temperature at the outlet first begins to rise and ends when the temperature is approaching steady state. The different simulation results correspond to simulations performed while varying only the mixing coefficient in PipeLumpedCap. Different values for the mixing coefficient resulted in different curves that modeled the experimental data.

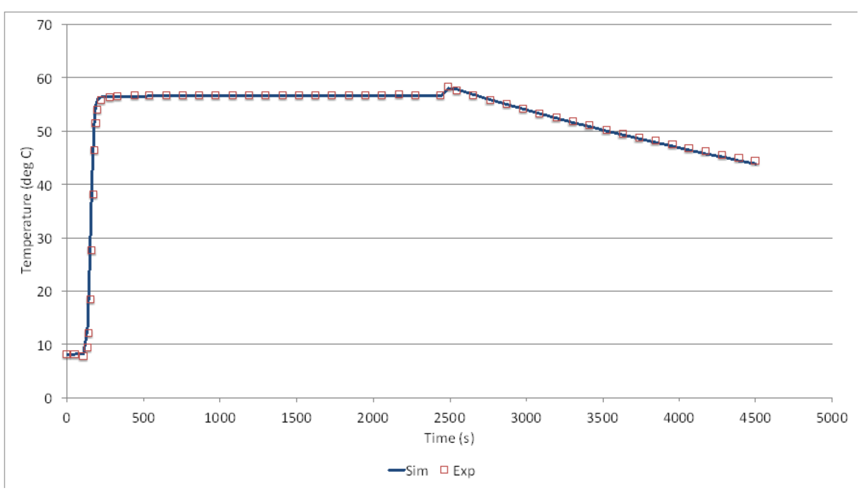
Based on the results in Figure 43 and the r^2 values shown in Table 13, 0.6 was selected for the mixing coefficient because it closely matched the entire curve. Because they closely match the top or the bottom of the curve, 0.7 or 0.5, respectively, might be more useful in some situations.

Table 13: R-squared Values for Results at Shown Mixing Coefficients

Mixing Coefficient	r^2
0.8	0.9942
0.7	0.9970
0.6	0.9984
0.5	0.9982
0.4	0.9960

Figure 44 shows what happens during a draw both in a laboratory and in a pipe simulated using the PipeLumpedCap component model. Data shown for both the simulation and experiment are the temperature of water exiting the pipe.

Figure 44: PipeLumpedCap Model Results Compared to Experimental Data



At the start of the test, the pipe is full of 8 °C (46.4 °F) cold water. The test is initiated by starting a hot water draw at 0.063 L/s (1 gal/min). The hot water pushes cold water out of the pipe and reaches the outlet of the pipe 135 seconds (2.25 min) into the test. The sudden temperature rise in both experimental and simulation data indicates the time when hot water reached the outlet of the pipe. The hot water flow ceases at 2,493 seconds (41.55 min). From 2,493 seconds

(41.55 min) to 4,498 seconds (74.97 min) the water remains stagnant in the pipe, gradually losing heat to the ambient environment.

The results in Figure 44 show excellent agreement between the simulation results and experimental data. The temperature rise as the hot water reaches the outlet of the pipe occurs at the same time with the same shape of the curve. The outlet temperature during the draw is nearly identical, as are the temperatures at the end of the pipe during the period of no-flow temperature decay at the end of the test.

These results are considered preliminary evidence that the model is valid. In all three tested conditions the simulation results closely matched the experimental data, implying that the model is accurate for the limited validated conditions. Future work will include performing validation comparisons at other flow rates, in other size pipes, and with more complex draw patterns.

5.1.2 End-Use Models

Describing the end uses of domestic hot water is necessary to complete the flow path from the mains water system through the hot water distribution pipes. The HWDS model contains components designed to describe generic end uses. The intent is that the user can apply the pipe models to create a distribution system, then the end-use models to describe the water flow rates. The generic model can be used to describe any end use when the user changes the inputs to describe a sink, shower, dishwasher, or other fixture.

Two generic end-use models are available to allow users to create simulations modeling both the hot and cold water pipes, and only the hot water pipes.

1. *EndUse*. This simple model specifies a mass flow rate and measures the outlet temperature. The user specifies a time-varying mass flow rate for the draw externally to this model, then passes it to the model through the watFlo input connector. The model provides a connection to the hot water distribution pipes and an outlet from the distribution system that provides the correct flow rate. The model also contains a temperature sensor for reporting the temperature of water leaving the piping system.

2. *EndUseTwoBranch*. This more complicated model is used to simulate both the hot and cold water distribution systems. Time-varying inputs for the total flow rate and the outlet temperature are used to describe a draw. Fluid connectors are used to connect the end-use model to both the hot and cold water pipes. The model contains a mixing valve and controller that together determine the proportion of hot and cold water flow to maintain the desired outlet temperature. An outlet from the system provides the correct total flow rate. A temperature sensor reports the temperature of water leaving the system.

The EndUseTwoBranch model represents a preliminary attempt to enable simulation of occupant behavior. This model allows the simulator to describe user behavior through the flow rate and temperature inputs. A shower can be simulated as a 0.1575 L/s (2.5 gal/min) draw at 38 °C (100 °F), for example. If the water temperature in a shower begins to decrease, the user will adjust the valves to maintain the desired temperature of the mixed hot water. The EndUseTwoBranch model models the same behavior by noticing the decreased hot water temperature and opening the mixing valve to allow more hot water flow, thus maintaining the desired outlet temperature. Before construction of this model, users had to arbitrarily increase the hot water flow rate in an attempt to emulate the adjustment without knowing the resulting outlet temperature.

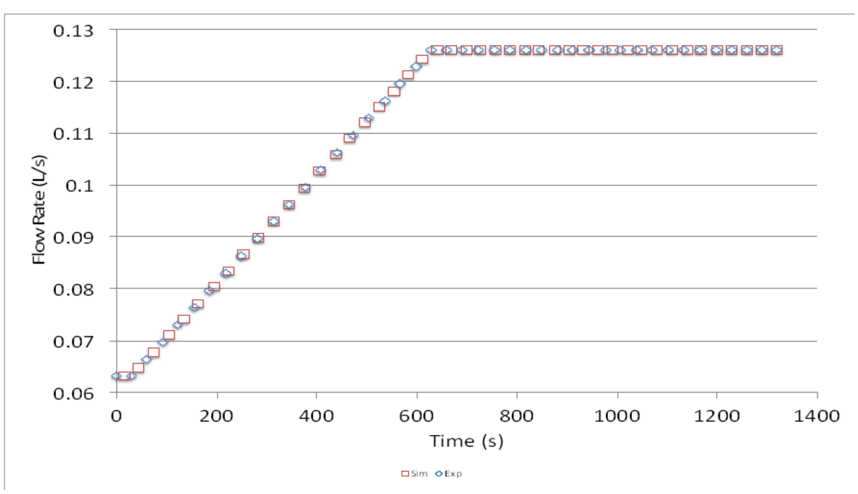
The default units for both end-use models are L/s. Alternative models allow specification of flow rate in gallons per minute. EndUseGPM is used for models examining only the hot water system. EndUseTwoBranchGPM is used to model both hot and cold water pipes.

5.1.3 Verification of End-Use Models

The end-use models are a means of controlling the outlet flow from the pipe models. Their function is to control the water flow rates in the pipes, not to provide thermo-fluid models of any specific component. The verification of the end-use models therefore focused solely on whether the end use correctly controlled water flows to achieve the desired outlet temperature and flow rate.

Simulations were performed to verify both EndUse and EndUseTwoBranch. The verification simulations connected the end-use models to a distribution system and checked to confirm that the simulated conditions in the system matched the desired conditions specified by the user. No experimental data were used for the verification process, only comparisons between the inputs specified by the user and the outputs the model predicted. Because the EndUseTwoBranch model is more complicated than the EndUse model, only verification results for EndUseTwoBranch are presented here. Figure 45 shows the desired and simulated flow rates in the EndUseTwoBranch verification simulation.

Figure 45: Desired and Simulated Water Flow Rates in the EndUseTwoBranch Verification Simulation



The desired draw flow rate at the start of the test was 0.063 L/s (1 gal/min). Thirty seconds into the simulation the desired flow rate began to slowly increase, ramping up to 0.126 L/s (2 gal/min) during the following 600 seconds (10 min). The water flow rate was held constant for the remaining 700 seconds (11.67 min) in the simulation as temperature conditions (shown in Figure 46) were changed.

The results in Figure 45 show that the output from the simulation closely matched the desired flow rate specified by the user.

Figure 46: Desired and Simulated Outlet Temperature in the EndUseTwoBranch Verification Simulation

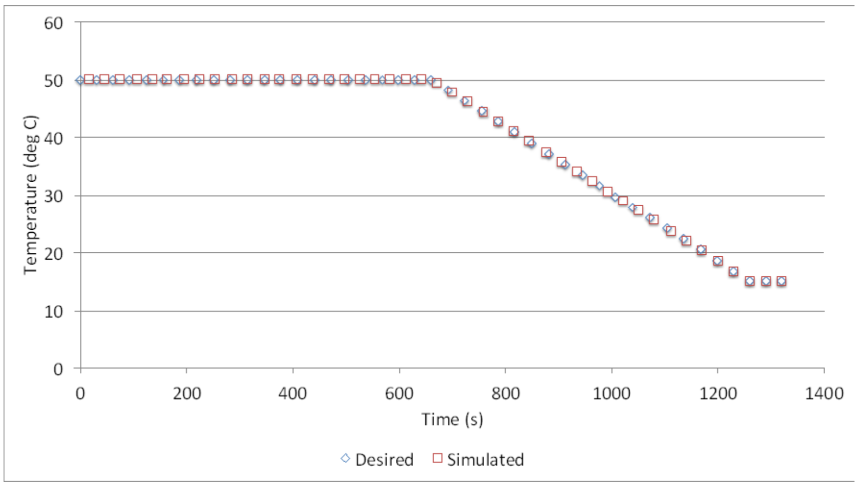


Figure 46 shows the desired and simulated outlet temperature during the EndUseTwoBranch verification simulation. The desired temperature leaving the fixture was 50 °C (122 °F) for the first 660 seconds (11 min) of the simulation. This part of the test was used to check the functioning of the flow rate control, as discussed regarding Figure 45. After 660 seconds (11 min), the desired outlet temperature began to decrease slowly. During the following 600 seconds (10 min), it decreased to 15 °C (59 °F), which was equal to the inlet cold water temperature. The desired outlet water temperature was held constant for the remaining 60 seconds (1 min) of the test.

Figure 46 shows that the output from the simulated end use closely matched the output requested in the simulation. Because the model is simple, and the results in Figures 45 and 46 matched well, the end-use models are considered ready to use.

5.1.4 FlowReduction Model

Lawrence Berkeley National Laboratory made an additional model component for controlling the flow rate of water. In situations of extremely high water flow, some tankless water heaters restrict the flow rate of water in order to continue to meet the temperature setpoint (Grant 2011). A FlowReduction model was created to determine the flow rate at each fixture during those events.

The FlowReduction model is placed between the desired hot water flow input file and the end-use models. The desired draw flow rate to each fixture is connected to the input of the FlowReduction model, and the outlet from FlowReduction (the actual draw flow rate) is connected to the end-use models.

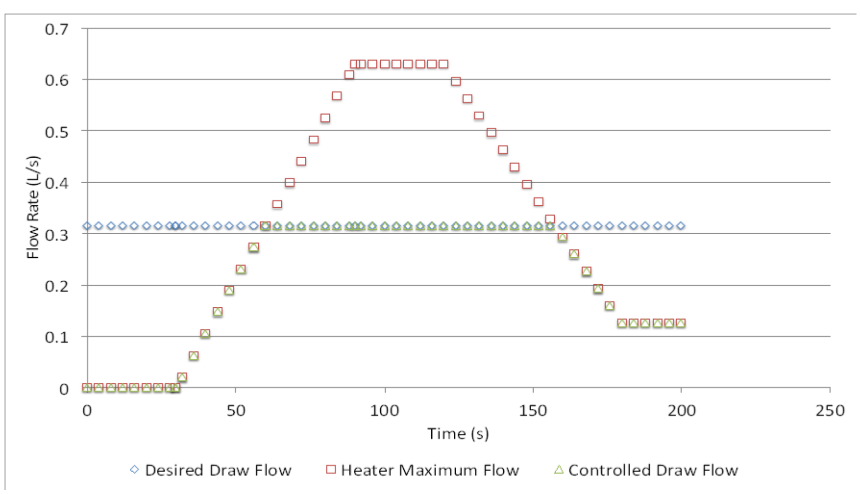
When the tankless water heater restricts the total water flow rate, the FlowReduction model adds the total desired water flow and compares that value to the flow allowed by the heater. When the total desired flow exceeds the flow allowed by the heater, FlowReduction reduces the flow of each draw by an equal percentage. The percentage is defined to make the total draw flow equal to the flow allowed by the heater.

Future research will focus on improving the piping models to more effectively model pressure drops in hot water distribution systems. When those modifications are complete, the flow rates for each draw in these situations will be determined based on the pressure drops in each pipe branch, and the FlowReduction model will be obsolete. FlowReduction is a temporary model used while those improvements are made. The FlowReduction model does nothing in cases when the total desired water flow is smaller than the flow allowed by the water heater.

5.1.5 Verification of FlowReduction Model

The FlowReduction model component was verified by checking its ability to reduce the total flow through a system. Because it restricts each draw individually, and the total system flow as a result, it was determined that if it correctly restricts one draw it will correctly reduce all active draws. The FlowReduction model was included in a simulation where a single draw surpassed the capacity of a tankless water heater. In the simulation the FlowReduction model was expected to reduce the draw flow rate to match the maximum flow rate of the tankless water heater. Figure 47 shows results from the simulation.

Figure 47: Desired, Heater Maximum, and Controlled Water Flow Rates in the FlowReduction Verification Simulation



The data in Figure 47 show a draw beginning at 30 seconds (0.5 min) and slowly increasing to 0.63 L/s (10 gal/min). The draw holds constant at 0.63 L/s (10 gal/min) for 30 seconds (0.5 min) before slowly decreasing to 0.126 L/s (2 gal/min). The heater maximum flow rate is constant at 0.315 L/s (5 gal/min). The controlled water flow rate follows the desired flow rate data until the desired flow rate surpasses the heater maximum flow rate 60 seconds (1 min) into the simulation. After 60 seconds (1 min), the controlled water flow rate follows the maximum water flow rate until 157.5 seconds (2.63 min) into the simulation. After 157.5 seconds (2.63 min), the desired water flow rate is again less than the heater's maximum flow rate and the controlled water flow matches the desired water flow for the rest of the simulation.

Because the results in Figure 47 show the FlowReduction model correctly controlling the draw flow, the model is considered ready for use.

5.1.6 Future Work

The current implementation of the HWDS library, although a major step toward providing complete simulation capabilities for residential hot water systems, could be improved in several ways. Future research efforts to improve the HWDS model are described below.

1. The current implementation of complex pipe models can cause simulations to perform slowly. The more complicated models contain many state events and non-linear equations, both of which are difficult for ODE solvers to handle. Improving the models to remove state events and non-linear equations will be a focus of future efforts.

2. Thus far efforts to develop the HWDS model have focused on the thermal side of the distribution system. However, the model has the capability to model pressure drop across the system. Important objectives of future research include validating the pressure drop across the pipe models, as well as creating models for other plumbing components such as elbows and Ts. Such improvements will enable simulators to determine the smallest-diameter pipe necessary to meet demand flow rates and to estimate the associated impact on energy consumption.

3. Further validation is necessary to confirm the correctness of the convection and radiation heat transfer rates in the PipeLumpedCap and PipeLumpedCapNoInsul models.

Chapter 6: Future Title 24 Revisions

In the past few years the average on-site energy use for heating water has exceeded that of space conditioning (KEMA-XENERGY, et al. 2005). Because water heating represents such a major energy end use, and is likely to increase further, it is worth examining the parts of California's Building Energy Efficiency Standards that pertain to water heating. Recently the Title 24 office held a staff workshop on scoping water heating systems for future building energy-efficiency standards.[4] This chapter, which supports that effort, explores how work being done at LBNL and elsewhere could be used to help shape future revisions California's building energy-efficiency standards.

6.1 Background

California's Building Energy Efficiency Standards (commonly known as Title 24 standards) address the building envelope, HVAC, water heating, and lighting (California Energy Commission 2013). The standards, originally adopted in 1978, have been updated periodically since. Title 24 establishes the requirements for calculating an energy budget for all new buildings, additions, and alterations. The energy budget varies by climate zone, building type, and building size. Any proposed building, addition, or alteration must be designed, built, and certified to meet the established energy budget. Compliance certificates attesting that the proposed design meets the budget must be submitted to the local building department. The building inspector (and sometimes a Home Energy Rating System [HERS] rater) certifies that the energy measures are installed correctly. Compliance can be prescriptive (by checklist) or performance-based (via computer modeling).

Title 24 standards are updated every three years to incorporate new energy-efficiency technologies, changes in the energy business, and state policy. The update process includes opportunities for public comment. The process of revising the 2016 Title 24 Standards already has begun. The investor-owned utilities in California will develop proposed changes and hold stakeholder meetings throughout the state in support of the 2016 standards update (Tam 2013). At the conclusion of the meetings, the Energy Commission will hold workshops in the spring of 2014 to present the draft standards. Any changes to the 2016 standards must be part of that review process.

The standards are based on the cost-effectiveness of energy-efficiency measures in new buildings in California. The standards promote measures that have a positive net present value of energy savings, construction costs, and maintenance costs during the life of the building. Since 2005 the cost-effectiveness calculations have relied on a time-dependent valuation (TDV).

---

[4] Future updates to the Building Energy Efficiency Standards. www.energy.ca.gov/title24/documents/.

The concept behind a TDV is that the value of energy-efficiency savings depends on which hours of the day and year the savings occur. Reflecting a combination of consumer costs and statewide societal costs, TDV is ultimately an economic metric and thus can be expressed in dollars (Price et al. 2011).

The legal requirements for meeting Title 24 building energy-efficiency standards are spelled out in the standards (California Energy Commission 2012). The water heating requirements are described in the Residential Compliance Manual (California Energy Commission 2013).

6.2 Calculations of Hot Water Use in Title 24

Unlike space conditioning energy use, which relies on computer simulation modeling, hourly water heating energy use is determined using equation-based calculations. The equations use the water heating system, the conditioned floor area, and the climate zone in which the building is located to determine hourly hot water energy use. A water heating system is defined by tank type; type of distribution system in the dwelling unit; presence of central water heating distribution (in multi-family buildings); efficiency (either energy factor or recovery efficiency with standby loss); tank volume; R-value of exterior insulation (for large storage or unfired water heaters); and rated input.

Water heating energy use is calculated for every hour of the year and then combined with the hourly space heating and cooling energy use to determine the total hourly fuel and electricity energy use, which then is factored by the hourly TDV energy multiplier. The methods and assumptions behind the calculations are presented in Brook et al. (2013), Appendix E. The calculation procedure applies to low-rise single-family, low-rise multi-family, and high-rise residential units.

6.2.1 Hourly Adjusted Recovery Load

The basis for the water heating calculation is the hourly adjusted recovery load (HARL), which is the heat content of the water delivered at the fixtures. The HARL is the hourly standard end use at all use points multiplied by a distribution loss multiplier. The HARL is adjusted, if appropriate, for savings provided by solar hot water heating, the hourly recirculation distribution losses associated with multi-family central water heating systems, and the tank surface losses of any unfired tank. The essential components of the equation for calculating HARL are shown below.

$$HARL = HSEU \times DLM + HRDL$$

Where:

HSEU = hourly standard end use at all use points (Btu)
DLM = distribution loss multiplier (unitless)
HRDL = hourly recirculation distribution loss (Btu)

HRDL occurs only for multi-family central water heating systems; it is zero for single-family dwellings.

6.2.2 Hourly Standard End Use

The hourly standard end use (HSEU) is the heat content of the water delivered at the fixture in terms of the draw volume in gallons per hour (GPH) times the temperature rise, ΔT (difference between the cold water inlet temperature and the hot water supply temperature), times the heat required to increase one gallon of water 1 °F (the 8.345 constant).

$$HSEU = 8.345 \times GPH \times \Delta T$$

Where:

GPH = hourly hot water consumption (gallons)
ΔT = temperature difference (°F)

6.2.2.1 Hourly Hot Water Consumption

The hourly hot water consumption (GPH) is calculated using the average daily hot water consumption and an hourly water consumption schedule.

$$GPH = GPD \times WF \times SCH$$

Where:

GPD = average daily hot water consumption (gallons) of the dwelling unit
WF = hot water waste factor
SCH = fractional daily load for each hour of the day

The hot water waste factor is 0.90 for recirculating systems within individual units or houses. Non-recirculating systems within individual units or houses and central systems in multi-family buildings are assigned a waste factor of 1.0.

The fractional daily load for each hour of the day (SCH) is taken from Table 14.

Table 14: Hourly Water Heating Schedules

Hour	Weekday	Weekend		Hour	Weekday	Weekend
1	0.014	0.018		13	0.036	0.051
2	0.008	0.01		14	0.033	0.043
3	0.009	0.009		15	0.032	0.039
4	0.011	0.008		16	0.026	0.039
5	0.02	0.015		17	0.042	0.052
6	0.044	0.023		18	0.048	0.058
7	0.089	0.026		19	0.052	0.056
8	0.107	0.047		20	0.047	0.052
9	0.089	0.077		21	0.042	0.047
10	0.066	0.083		22	0.039	0.044
11	0.052	0.074		23	0.036	0.04
12	0.038	0.061		24	0.022	0.028
				Sum	1	1

6.2.2.2 Daily Hot Water Consumption

The average daily hot water consumed at use points is equal to 21.4 gallons per day (gpd) plus an additional 6.79 gpd for each 1,000 square feet (ft^2) of conditioned floor area. Consumption is equal to 26.2 gpd for a 700 ft^2 apartment and 38.4 gpd for a 2,500 ft^2 dwelling unit. The equation for daily hot water consumption is expressed as follows.

$$\begin{aligned} GPD &= 21.4 + 0.00679 \times CFA \text{ if } CFA \leq 2{,}500\text{, and} \\ &= 38.375 \qquad\qquad\qquad \text{if } CFA > 2{,}500 \end{aligned}$$

6.2.2.3 Temperature Difference

The hourly temperature difference between the cold inlet water and the hot water supply (ΔT) is calculated as follows:

$$\Delta T = Ts - Tinlet$$

Where:

Ts = hot water supply temperature (°F)

Tinlet = cold water inlet temperature (°F)

The hot water supply temperature is set to 54 °C (130 °F) for multi-family buildings that have central recirculation loops; otherwise, it is set to 51 °C (124 °F).

6.2.2.4 Water Heater Inlet Temperature

The water heater inlet temperature (*Tinlet*) is based on the climate zone. It is assumed to vary daily based on the following relationship defined by data in the climate zone weather files. For each hour of the year, *Tinlet* is calculated as follows:

$$Tinlet = Tground \times 0.65 + Tavg_{31} \times 0.35$$

Where:

$Tavg_{31}$ is the dry-bulb temperature averaged over the previous 31 days (for January days, weather data from December is used)

Tground is the hourly ground temperature

Tground for each hour of the year (θ) is calculated by the following equation:

$$Tground_{\theta} = Tyr_{ave} - 0.5 \times (Tyr_{max} - Tyr_{min}) \times \cos(2\pi\theta/8760 - 0.6 - \varphi) \times \Theta$$

Where:

Tyr_{ave} = average annual temperature (°F)

Tyr_{max} = the highest average monthly temperature (°F)

Tyr_{min} = the lowest average monthly temperature (°F)

8,760 = number of hours in a year

$$\beta = \sqrt{(\pi/(0.0435/8760))} \times 10 = 7953.941$$

$$\varphi = \text{atan}(\,(1 - e^{-\beta}(\cos(\beta) + \sin(\beta))\,/(1 - e^{-\beta}(\cos(\beta) - \sin(\beta))\,) = 0.785398$$

$$\Theta = \sqrt{(((e^{-\beta})^2 - 2 \times e^{-\beta} \times \cos(\beta) + 1\,)/(2 \times \beta^2))} = 0.000088900018$$

6.2.3 Distribution Loss Multiplier

Hourly standard end use is multiplied by the distribution loss multiplier (DLM) to calculate the hourly adjusted recovery load. The DLM accounts for heat loss attributable to the hot water distribution system. The DLM combines two terms: the standard distribution loss multiplier (SDLM), which depends on the floor area of the dwelling unit, and the distribution system multiplier (DSM). The DSM is an adjustment for alternative water heating distribution systems within a dwelling unit.

$$DLM = 1 + (SDLM - 1) \times DSM$$

Where:

SDLM = standard distribution loss multiplier (unitless)

DSM = distribution system multiplier (unitless)

Standard Distribution Loss Multiplier

The standard distribution loss multiplier (SDLM) for a dwelling unit is calculated as follows. The SDLM is capped at 2,500 ft² for all single- and multi-family units.

$$SDLM = 1.004 + 0.000202 \times CFA - 0.0000000231 \times CFA^2 \quad \text{if } CFA \leq 2{,}500 \text{, and}$$

$$= 1.364625 \quad \text{if } CFA > 2{,}500$$

6.2.3.1 Distribution System Multiplier

The distribution system multiplier (DSM) is a unitless adjustment for alternative water heating distribution systems in a dwelling unit. A value of 1.00 is used for standard distribution systems. Standard distribution systems are defined as non-recirculating systems in which all pipes of a nominal ¾" or larger diameter, and the full length of the line from the water heater to the kitchen fixtures are insulated to a nominal R-4.

Assigned DSM values for various distribution system types within a dwelling unit are shown in Table 15. Improved DSM values are available for cases where voluntary HERS inspections are performed.

Table 15: Distribution System Multipliers for a Dwelling Unit

Distribution System Type	Assigned DSM
No HERS Inspection Required	
Trunk and Branch—Standard (STD)	1.0
Pipe Insulation (PIC)	0.9
Parallel Piping (PP)	1.05
Recirculating: Non-Demand Control Options (R-ND)	7.0
Recirculating with Manual Demand Control (RDRmc)	1.15
Recirculating with Motion Sensor Demand Control (R-DRsc)	1.3
Optional Cases: HERS Inspection Required	
Pipe Insulation (PIC-H)	0.8
Parallel Piping with 5' maximum length (PP-H)	0.95
Compact Design (CHWDS-H)	0.7
Point of Use (POU-H)	0.3
Recirculating with Manual Demand Control (RDRmc-H)	1.05
Recirculating with Motion Sensor Demand Control (RDRsc-H)	1.2

6.2.4 Hourly Recirculation Distribution Loss for Central Water Heating Systems

The hourly recirculation distribution loss (HRDL) applies to the recirculation loop for multi-family central water heating systems. The DLM accounts for the distribution heat loss within each individual dwelling unit. Additional distribution losses that occur outside dwelling units include losses from recirculation loop pipes and branch piping that feeds the individual units.

The calculations for HRDL were updated extensively in the 2013 revision of the Title 24 standards (Zhang 2011 and 2013). The calculations are only summarized here.

The hourly values of the HRDL is calculated as follows.

$$HRDL = NLoop \times HRLL + HRBL$$

Where:

$NLoop$ = number of recirculation loops in the water heating system (applicable only if the system has multiple recirculation loops)

$HRLL$ = hourly pipe heat loss in the recirculation loop (Btu)

$HRBL$ = hourly heat loss in the recirculation branch pipes (Btu)

6.2.4.1 Hourly Heat Loss for Recirculation Loop Pipe

Hourly recirculation loop loss (HRLL) is the hourly heat loss from all sections of pipe involved in recirculation. Pipe heat losses include those associated with water flow, and those that occur without hot water flow. The latter happens when the control system turns off the recirculation pump and there are no hot water draw flows, such as in the recirculation return pipes. Pipe heat loss modes are determined by the recirculation control schedules and the hot water draw schedules. For each pipe section, hourly pipe heat loss is the sum of heat loss from the two modes.

6.2.4.2 Hourly Heat Loss for Recirculation Branch Pipe

Branch pipe heat loss has two components. First, pipe heat losses occur when hot water is in use. Second, there could be losses associated with hot water waste when (a) hot water is used to displace cold water in branch pipes, and (b) hot water is left in the pipe after hot water draws and is so cool it must be dumped down the drain. The hourly recirculation branch losses include both components.

6.3 Improving the Calculation of Hot Water Energy Use

The manner in which water heating energy use is calculated for energy-efficiency building standards has several shortcomings. Given the complexity and variety of hot water distribution system configurations and the variability in the ways people use hot water, it is necessary to make simplifying assumptions.

The type of assumptions and the level of detail in the assumptions in the hourly adjusted recovery load often exceed the understanding of hot water use in field conditions. For example, the cold water inlet temperature is calculated from a complicated equation of ground temperature that uses data from the climate zone weather files. Cold water inlet temperatures have been measured in several field-monitoring studies. These field data should be used to evaluate, and perhaps simplify, the calculation of cold water inlet temperatures.

Another example is the hourly hot water usage schedules. Those schedules show hot water being used every hour of the day. The calculation of hourly average recovery load converts this use schedule to using water heating energy every hour of the day. Storage water heaters heat water for only a short period of time each day. The hot, stored water meets most of the hot water demands. This buffering effect of storage water heaters means the time of day of energy consumption, and thus the TDV, is not calculated accurately. This effect is important when considering heat pump water heaters and other efficient electric water heaters.

Another way to evaluate the calculations would be to do an uncertainty analysis on the algorithms used in the calculations. This analysis could be done by systematically varying the inputs to each of the terms in the calculation and assessing the impact on the final result. The terms that have the most impact when varied should be examined more closely. It may be possible to rewrite the calculations to make the assumptions and results more robust.

Recent field-monitoring data on hot water use and water heating energy consumption should be compared with results of water heating energy calculations done according to the standard for that building. This comparison could be done to verify, and correct as necessary, the current calculation methods.

6.4 Simulation Models

The Energy Commission has used simulation models to support the development of the water heating part of Title 24. The primary model has been HWSIM, a hot water distribution simulation program that models the heat losses through a piping network from the water heater to each hot water end-use point in a home (Springer et al. 2008).

Recently LBNL developed the collection of water heating models described in this document. These models could be used to aid in the design of future versions of Title 24. The models for non-condensing storage tank and non-condensing tankless water heaters were designed to be flexible and easily adapted to match current water heaters. Both models are based on a few parameters that describe the water heater, enabling the user to emulate different manufacturers' products by changing only a few inputs. This flexibility creates the opportunity for researchers to quantify the change in energy consumption based on the selected water heater.

The models for the hot water distribution system (HWDS) incorporate a modular design to facilitate creation of various HWDS. The collection of HWDS models incorporate various ways of representing pipes and end uses such as sinks, showers, or dishwashers. The models vary in complexity, allowing the user to choose between highly detailed modeling or rapid results.

Models are designed so that users can combine models easily to create a whole-building hot water simulation. The user can represent the connection to the mains water supply, the water heater, the hot and cold water distribution pipes, and the end uses by simply dragging, dropping, and connecting the appropriate components. This design method enables users to quickly create a wide range of simulations, representing the highly varied designs of hot water systems in California buildings. Water use profiles can be read from text files, enabling the user to set up repeated simulations using a different draw profile each time.

Some of the water heater models in the LBNL collection have been validated using experimental data. The non-condensing storage tank and non-condensing tankless water heater models were validated using experimental data collected at LBNL. The data, which were collected from experiments designed specifically to validate the models, are considered sufficient. The HWDS models, on the other hand, were validated using a limited quantity of data from laboratory experiments on pipe heat loss. Although the HWDS models are a valuable simulation tool, more validation should be performed before they are adopted for widespread use.

Whereas the laboratory validation process demonstrated that the LBNL models accurately emulate the physics in water heating systems, the results of simulations have not been compared to field data. Future efforts should include comparing the results of simulations to results of data collected in field studies, and seeking to identify and remedy any discrepancies.

Currently the HWDS models have been validated only for use in thermal simulations. The capability of the models to identify pressure drop and maximum fluid flows through the pipes has not yet been validated. Further validation must be performed before the models are used to answer questions relating to maximum water flow, pressure drop, and pipe diameter. There are limited published data regarding pressure drops through plastic pipes and fittings. Further experimentation is needed before the HWDS simulation models can be validated for applications that utilize plastic pipes.

6.5 Installation Issues

Current plumbing practice does not commonly address energy efficiency. The construction industry may not readily accept changes. In response to concerns from the construction industry, the Energy Commission has withdrawn recommendations to adopt new requirements for showers and limits on the length of 1" hot water piping (Brook et al. 2013).

The building envelope is well planned before construction begins. Those plans are used to build the building energy simulation models that generate the energy budgets used to demonstrate compliance with Title 24. The layout of the pipes for hot water distribution systems are not included in the plans for new residential construction. It is impossible for a simulation model of the hot water distribution system to correctly predict the performance of something that is not designed until it is being built. Until this situation changes, it is unreasonable to expect building energy simulation models to include water heating energy use.

Enforceability of energy efficiency regulations of hot water systems will require establishing methods for determining compliance that can be easily conducted in the field. For example, building officials would not want to measure the length and diameter of every piece of hot water pipe during a building inspection. Although ease of enforceability in the field is not an issue with regard to hot water energy use calculations, it must be accounted for to assure the successful adoption of standards.

GLOSSARY

Term	**Definition**
ASHRAE	American Society of Heating and Air-Conditioning Engineers
Btu	British thermal unit
buoHeaTra	buoyant flow heat transfer
cirHeaTra	circulation heat transfer
cm	centimeter
CO_2	carbon dioxide
deg	degrees
DLM	distribution loss multiplier
draHeaTra	draw heat transfer
DSM	distribution system multiplier
FSEC	Florida Solar Energy Center
ft^3	cubic foot
gal	gallons
GPD	gallons per day
GPH	gallons per hour
HARL	hourly adjusted recovery load
HEATER	U.S. DOE water model
HERS	Home Energy Rating System
HRDL	hourly recirculation distribution loss
$hr\text{-}ft^2\text{-}F$	hour square foot degree Fahrenheit
HRLL	hourly recirculation loop loss
HSEU	hourly standard end use
HVAC	heating, ventilation, and air conditioning
HWDS	hot water distribution systems
HWSIM	California Energy Commission water heating system model

in	inch
J	joules
kBtu	thousand British thermal units
kg-K	kilograms Kelvin
kg/m^3	kilograms per cubic meter
kW	kilowatt
L/s	liter per second
lb-F	pound degree Fahrenheit
LBNL	Lawrence Berkeley National Laboratory
m-K	meter Kelvin
min	minute
m^3	cubic meter
NAHB	National Association of Home Builders
ODE	ordinary differential equation
ORNL	Oak Ridge National Laboratory
PID	proportional, integral, and derivative values
PIER	Public Interest Energy Research
POU	point-of-use
RD&D	research, development, and demonstration
SCH	fractional daily load for each hour of the day
SDLM	standard distribution loss multiplier
TANK	Gas Research Institute water model
TDV	time-dependent valuation
TRNSYS	Solar Energy Lab (University of Wisconsin) water model
UA	a heat transfer coefficient
uALos	UA heat loss
U.S. DOE	U.S. Department of Energy

WATSIM	Electric Power Research Institute water model
WF	waste factor

REFERENCES

ADL. 1982a. *Engineering Computer Models for Refrigerators, Freezers, Furnaces, Water Heaters, Room and Central Air Conditioners.* Arthur D. Little, Cambridge, Mass. No. 83528-20 under DE-AC01-78CS20420, November.

ADL. 1982b. *Water Heater Computer Model User's Guide.* Arthur D. Little, Cambridge, Mass. DE-AC01-78CS20420, November.

Baskin, E., W. G. Craddick, R. Lenarduzzi, R. L. Wendt, and K. A. Woodbury. 2007. *Hot Water Distribution System Program Documentation and Comparison to Experimental Data.* Oak Ridge National Laboratory. Provided to the Energy Commission. September. http://info.ornl.gov/sites/publications/files/Pub8207.pdf.

Bradley, D., and M. Kummert. 2005. "New Evolutions in TRNSYS – a Selection of Version 16 Features." In B*uilding Simulation 2005* Montréal. 107–114. http://www.ibpsa.org/proceedings/BS2005/BS05_0107_114.pdf.

Brook, M., D. Herr, J. Miller, D. A. Ross, and D. Tam. 2013. *2013 Residential Alternative Calculation Method Reference Manual - Revised Final Draft.* California Energy Commission. http://www.energy.ca.gov/2013publications/CEC-400-2013-003/CEC-400-2013-003-SD-REV.pdf.

Burch, J., J. Thornton, M. Hoeschele, D. Springer, and A. Rudd. 2008. "Preliminary Modeling, Testing, and Analysis of a Gas Tankless Water Heater." In *SOLAR 2008* San Diego, http://www.nrel.gov/docs/fy08osti/42917.pdf.

California Energy Commission. 2012. 2013 Building Energy Efficiency Standards, Title 24, Part 6, and Associated Administrative Regulations in Part 1." California Energy Commission. http://www.energy.ca.gov/title24/2013standards/rulemaking/documents/final_rulemaking_documents/44_Final_Express_Terms/2013_Standards_FINAL.pdf.

California Energy Commission. 2013. *Residential Compliance Manual for the 2013 Building Energy Efficiency Standards, Title 24, Part 6, and Associated Administrative Regulations in Part 1.* California Energy Commission. http://www.energy.ca.gov/2013publications/CEC-400-2013-001/CEC-400-2013-001-SD.pdf.

Consul, R., I. Rodriguez, C. D. Perez-Segarra, and M. Soria. 2004. "Virtual prototyping of storage tank by means of three-dimensional CFD and heat transfer numerical simulations." *Solar Energy* 77: 179–191. http://www.sciencedirect.com/science/article/pii/S0038092X04000842.

Cragen, K. E. 1994. "Impact on a Utility of an Ensemble of Solar Domestic Hot Water Systems." Masters Thesis. Madison: University of Wisconsin. p. 254. http://minds.wisconsin.edu/handle/1793/7857.

Davis Energy Group Inc., and E. Inc. 1990. "Residential Water Heating Study: Technical Report: Approved Draft." Davis Energy Group, Davis, California. Report. May 23.

Davis Energy Group, and R. Scott. 2007. "HWSIM Hot Water Distribution Model Validation Report." Davis Energy Group, Subcontract #: 6803947. May 25.

Dynasim. 2009. "Dymola Multi-Engineering Modeling and Simulation." Lund, Sweden: Dynasim AB.

Ellul, M. K., and A. Muscat. 2008. "Computer Modelling and Simulation of a Smart Water Heater." In *Workshop in Information and Communication Technology.* University of Malta. http://www.um.edu.mt/__data/assets/pdf_file/0006/51765/wict08_submission_30.pdf.

Elmqvist, H., H. Olsson, S. E. Mattson, D. Bruck, C. Schweiger, D. Joos, and M. Otter. 2005. "Optimization for Design and Parameter Estimation." Modelica. Ed. Gerhard Schmitz. Hamburg, Germany. 255–266.

Fujii, T., M. Takeuchi, M. Fujii, K. Suzaki, and H. Uehara. 1970 . "Experiments on Natural-convection Heat Transfer from the Outer Surface of a Vertical Cylinder to Liquids." *International Journal of Heat and Mass Transfer* 13(5) (May): 753–787. doi:10.1016/0017-9310(70)90125-0.

Goldschmidt, V. W., and C. Beckermann. 1986. "Heat Transfer Augmentation in the Flueway of a Water Heater." *ASHRAE Transactions* 92: 485–495.

Goldschmidt, V. W. and C. C. Offhaus. 1987. "Performance of Different Types of Flueway Baffles in a Gas-fired Water Heater. *ASHRAE Transactions* 93: 1463–1473.

Goldschmidt, V. W., C. C. Offhaus, and C. Beckermann. 1989. "Effect of the Flueway Battle Emissivity on the Performance of a Gas-Fired Water Heater." *ASHRAE Transactions* 95: 281–283.

Grant, P. "Evaluation of the Performance of Gas Tankless Water Heaters." 2011. Thesis submitted for the Degree of Master of Science. Boulder, Colorado: University of Colorado at Boulder, Department of Civil, Environmental and Architectural Engineering.

Grant, P., J. Burch, and M. Krarti. 2010. "Modeling Gas Tankless Water Heaters and Studying the Impacts of Draw Profile on System Efficiency." National Renewable Energy Laboratory

Gu, L. "A Simplified Hot Water Distribution System Model." 2007. In *International Building Performance Simulation Association*. Beijing, China. 8. http://www.fsec.ucf.edu/en/publications/pdf/FSEC-PF-429-07.pdf.

Guo, J. J., J. Y. Wu, R. Z. Wang, and S. Li. 2011. "Experimental research and operation optimization of an air-source heat pump water heater." *Applied Energy* 88: 4128–4138. November. http://www.sciencedirect.com/science/article/pii/S0306261911002340.

Hiller, C. C. 2006a. "Hot Water Distribution System Piping Time, Water, and Energy Waste-Phase I: Test Results." *ASHRAE Transactions* 112: 415–425. http://www.techstreet.com/cgi-bin/detail?product_id=1718782.

Hiller, C. C. 2006b. "Hot Water Distribution System Piping Heat Loss Factors- Phase I: Test Results." *ASHRAE Transactions*, 112: 436–446. http://www.techstreet.com/cgi-bin/detail?product_id=1718783.

Hiller, C. C. 2008a. "Hot Water Distribution System Piping Heat Loss Factors, Both In Air and Buried - Phase II Test Results." *ASHRAE Transactions* 114. http://www.techstreet.com/cgi-bin/detail?product_id=1715429.

Hiller, C. C. 2008b. "Hot Water Distribution System Piping Time, Water, and Energy Waste - Phase II Test Results." *ASHRAE Transactions* 114. http://www.techstreet.com/cgi-bin/detail?product_id=1712122.

Hiller, C. C. 2011a. "Hot-Water Distribution System Piping Time, Water, and Energy Waste—Phase III: Test Results." *ASHRAE Transactions* 117. http://www.techstreet.com/cgi-bin/detail?product_id=1801416.

Hiller, C. C. 2011b. "Hot-Water Distribution System Piping Heat Loss Factors—Phase III: Test Results." *ASHRAE Transactions* 117. http://www.techstreet.com/cgi-bin/detail?product_id=1801456.

Hiller, C. C., A. I. Lowenstein, and R. L. Merriam. 1994. "Detailed Water Heating Simulation Model." In *ASHRAE Transactions*. New Orleans.

Hmouda, I., I. Rodriguez, C. Bouden, and A. Oliva. 2010. "Unsteady natural convection cooling of a water storage tank with an internal gas flue." *International Journal of Thermal Sciences* 49: 36–47. January. http://www.sciencedirect.com/science/article/B6VT1-4WR5NVC-1/2/e1ae162bfe910be676ee0df769c9e6d3.

Hoover, W., PhD. 2012. Personal Conversation. ACEEE Hot Water Forum. May 21.

Hoeschele, M., and E. Weitzel. 2012. "Hot Water Distribution System Model Enhancements." Davis, California: Alliance for Residential Building Innovation (ARBI) Davis Energy Group, Team Lead for U.S. Department of Energy's Building America Program. November http://arbi.davisenergy.com/?publication=hot-water-distribution-model-enhancements&wppa_download=1.

Idem, S. A., A. M. Jacobi, G. M. Maxwell, and V. W. Goldschmidt. 1992. "A Performance Model of an Instantaneous, Condensing, Gas-Fired Hot Water Boiler." *ASHRAE Transactions* 98 (Pt. 2): 165–171. http://www.techstreet.com/cgi-bin/detail?product_id=1716731.

KEMA-XENERGY, Itron, and RoperASW. 2005. "2003 California Statewide Residential Appliance Saturation Study, Final Report Executive Summary." 400-04-009. http://www.energy.ca.gov/reports/400-04-009/2004-08-17_400-04-009ES.PDF.

Klein, G. 2006. "Hot Water Distribution Research." *Official* 39–44. September/October. http://www.cuwcc.org/res_hot_water/HotWaterDist_Part4-Klein.pdf.

Kleinbach, E. M., W. A. Beckman, and S. A. Klein. 1993. "Performance study of one-dimensional models for stratified thermal storage tanks." *Solar Energy* 50: 155–166.

http://www.sciencedirect.com/science/article/B6V50-497T94P-JW/2/837750211617c8b33d5578857b277641.

Kosar, D., P. Glanville, and H. Vadnal. 2013. *Residential Water Heating Program: Facilitating the Market Transformation to Higher Efficiency Gas-Fired Water Heating.* Sacramento: Gas Technology Institute for the California Energy Commission.

Li, B. 2011. "High Resolution Time-Series Modeling of Domestic Hot Water Heating Systems." Master of Applied Science. Department of Mechanical Engineering. Victoria, BC: University of Victoria. p. 132. https://dspace.library.uvic.ca:8443/bitstream/handle/1828/3611/Master%20Thesis_Final_PDF.pdf.

Lutz, J. 2010. *Evaluation of TANK water heater simulation model as embedded in HWSim.* Lawrence Berkeley National Laboratory. December 22.

Lutz, J. 2012. "Water and Energy Wasted During Residential Shower Events: Findings from a Pilot Field Study of Hot Water Distribution Systems." In *ASHRAE Transactions*. Chicago.

MacGregor, R. K., and A. F. Emery. 1969. "Free Convection Through Vertical Plane Layers—Moderate and High Prandtl Number Fluids." *Journal of Heat Transfer* 91: 11. August. http://heattransfer.asmedigitalcollection.asme.org/data/Journals/JHTRAO/27344/391_1.pdf.

Mohamed, A. E. 1994. "Experimental investigation of natural convection from vertical helical coiled tubes." *International Journal of Heat and Mass Transfer* 37: 665–671. http://www.sciencedirect.com/science/article/pii/0017931094901384.

Parker, M. 2011. Plenary Presentation to ACEEE Hot Water Forum. May. http://www.aceee.org/files/pdf/conferences/hwf/2011/Plenary%20-%20Mike%20Parker.pdf.

Paul, D. D., G. R. Whitacre, J. J. Crisafulli, R. D. Fischer, A. L. Rutz, J. G. Murray, and S. G. Holderbaum. 1993. *TANK Computer Program User's Manual with Diskettes: An Interactive Personal Computer Program to Aid in the Design and Analysis of Storage-Type Water Heaters.* Battelle Memorial Institute GRI-93/0186. July.

Price, S., A. Mahone, N. Schlag, and D. Suyeyasu. 2011. *Time Dependent Valuation of Energy for Developing Building Efficiency Standards: 2013 Time Dependent Valuation (TDV) Data 4s and Inputs.* San Francisco: Energy and Environmental Economics, Inc. http://www.energy.ca.gov/title24/2013standards/prerulemaking/documents/general_cec_documents/Title24_2013_TDV_Methodology_Report_23Feb2011.pdf.

Rheem. RTG84-DVLN specifications sheet. No date. http://www.rheem.com/product/tankless-mid-efficiency-tankless-mid-efficiency-84-direct-vent-indoor.

Ryan, David, Ray Long, Daniel Lauf, Meredith Ledbetter, and Ari Reeves. 2010. "ENERGY STAR Water Heater Market Profile: Efficiency Sells." U.S. Department of Energy.

September. http://www.energystar.gov/ia/partners/prod_development/new_specs/downloads/water_heaters/Water_Heater_Market_Profile_2010.pdf.

Schneyer, C. D. 2011. "Simulation of energy use in residential water heating systems." Master of Applied Science. Department of Mechanical Engineering. Victoria, BC: University of Victoria. p. 98. https://dspace.library.uvic.ca:8443/bitstream/handle/1828/3526/Schneyer_Carolyn_MASc_2011.pdf.

Schultz, W. W., and V. W. 1983. Goldschmidt. "Distribution Efficiency and Characteristics of Start-up and Shutdown of Flow in Hot Water Distribution Lines." *ASHRAE Transactions* 89: 42–60.

Springer, D., L. Rainer, M. Hoeschele, and R. Scott. 2008. "HWSIM: Development and Validation of a Residential Hot Water Distribution System Model." In *Summer Study on Energy Efficiency in Buildings.* Asilomar, California. 267–277. www.aceee.org/sites/default/files/publications/proceedings/SS08_Panel1_Paper21.pdf.

Tam, D. 2013. "Title 24 Rulemaking Overview" presented at the Scoping Water Heating Systems Workshop, July 16, Sacramento, California. http://www.energy.ca.gov/title24/documents/2013-07-16_workshop/presentations/Scoping_Water_Heating_systems_Workshop.pdf.

Thermal Energy System Specialists. 2010. "TESS Component Libraries General Descriptions." http://www.trnsys.com/tess-libraries/TESSLibs17_General_Descriptions.pdf.

Tiller, M. 2001. "Introduction to Physical Modeling with Modelica." The Springer International Series in *Engineering and Computer Science.*

U.S. Department of Energy-Office of Conservation and Renewable Energy. 1998. "Energy Conservation Program for Consumer Products: Test Procedures for Water Heaters; Final Rule." *Federal Register* 63: 25996–26016. May 11. http://www.eere.energy.gov/buildings/appliance_standards/residential/pdfs/wtrhtr.pdf.

U.S. Department of Energy-Office of Energy Efficiency and Renewable Energy. 2001. "Energy Conservation Program for Consumer Products: Energy Conservation Standards for Water Heaters; Final Rule." *Federal Register* 66: 4474–4497. January 17.

University of Wisconsin-Madison College of Engineering's Solar Energy Lab. 2011. "A TRaNsient SYstems Simulation Program, TRNYSYS 17." http://sel.me.wisc.edu/trnsys/index.html.

Valencia, M. P. P. a. 2004. "Condensation of Water Vapor and Acid Mixtures from Exhaust Gases." Institut für Energietechnik Fachgebiet Energie-, Impuls- und Stofftransport. PhD. Berlin: Technischen Universität Berlin. http://opus.kobv.de/tuberlin/volltexte/2005/794/pdf/plaperujo_maria.pdf.

Wetter, M., W. Zuo, and T. Nouidou. 2011. "Recent Development of the Modelica Buildings Library for Building Heating, Ventilation and Air-Conditioning Systems." *Proceedings of the 8th International Modelica Conference*. Technical University, Dresden, Germany, March 20–22.

Wiehagen, J., and J. L. Sikora. 2002. "Domestic Hot Water System Modeling for the Design of Energy Efficient Systems." NAHB Research Center, Upper Marlboro. April 2. http://www.toolbase.org/PDF/DesignGuides/domestichotwater.pdf#search=%22domestic%20hot%20water%20system%20modeling%20for%20the%20design%20of%20energy%20efficient%20systems%22.

Wendt, R., E. Baskin, and D. Durfee. 2004. *Final Report: Evaluation of Hot Water Distribution Systems by Numerical Simulation.* Oak Ridge National Laboratory. March. http://www.ornl.gov/~webworks/cppr/y2001/rpt/122464.pdf.

Zhang, Y. 2011. "Nonresidential Space and Water Heating Proposal, California Statewide Utility Codes and Standards Program." May 24. http://www.energy.ca.gov/title24/2013standards/prerulemaking/documents/2011-05-24_workshop/presentations/NR_Water_and_Space_Heating_CEC_Workshop_May_24_2011.pdf.

Zhang, Y. 2013. *Multi-family Central Domestic Hot Water Distribution Systems.* Final Project Report. CEC-500- 2013-011. Heschong Mahone Group, Inc. for the California Energy Commission. http://www.energy.ca.gov/2013publications/CEC-500-2013-011/CEC-500-2013-011.pdf.

Zurigat, Y. H., A. J. Ghajar, and P. M. Moretti. "Stratified thermal storage tank inlet mixing characterization." *Applied Energy* 30: 99–111. 1988. http://www.sciencedirect.com/science/article/pii/0306261988900074.