ABSTRACT

An iterative mathematical technique utilizing finite element modeling was used to simulate and study beer cooling in a vertical cylindroconical fermenter. The model system assumed a geometry that typified most brewery tank designs. A theoretical single-point measurement position for the tank mean temperature is presented and is consistent with existing field data.

Keywords: fluid mechanics, finite element analysis, simulation, fermentation, cooling, cylindroconical

INTRODUCTION

Brewers pride themselves in providing a consistent, quality product to the consumer. As product curators, they employ the latest biological, engineering, and raw materials technologies to ensure brand equity while remaining competitive in the marketplace. Of the many processes under the brewer’s control, one unit operation of particular interest is cooling of end-fermented beer in a vertically oriented cylindroconical vessel (CCY). During this procedure, warm fermented beer is chilled and stored within a narrow temperature range. The brewer’s specific objectives are: (a) cooling the beer quickly to curtail fermentation before filtration/centrifugation or (b) storing the beer at a single average temperature to ensure proper development of aroma and flavor.

With increasing scale, control of CCY cooling has proven to be an elusive design problem for both brewers and tank design engineers, who have developed only empirical guidelines to assist their processing efforts. If the unit operation could be modeled mathematically, a better understanding of the system would be gained, resulting in better process design, product consistency, and higher quality.

BACKGROUND

In the early 1900s, several geometries for larger scale beer production were in existence, such as cubes, cylinders, slanted tanks, spheres, or other vats of various shapes. Many vessels were additionally fitted with cooling jackets, recirculation pumps, or gas injection equipment in an effort to produce greater quantities of consistent quality beer. By the mid-1930s, the Nathan group of Germany introduced a CCY (called a Nathan tank) that could be used for fermentation or aging of beer. This style of tank provided benefits to the brewer and engineer over previous designs, and rectified many of the processing problems.
trade-offs that hampered quality and productivity. A few of these advantages were: (a) a vertically oriented vessel gave a higher output of product per area in the plant, (b) flocculent yeast was conveniently recovered through the conical bottom of the tank, (c) beer loss was reduced since fermentation and aging was completed in a single vessel, and (d) CO₂ evolved from the process could be collected and reused for carbonation adjustment.

Early evidence of CCV fluid flow study was presented in 1938 when Bishop noted the mixing characteristics of yeast suspensions during fermenter cooling. Few additional articles were published until a landmark paper was presented by Delente, Akin, Krabbé, and Ladenburg in 1968. In their trials, cross-sectional cutaway tanks closed with Plexiglas® were constructed, and beer was prepared in the vessels to monitor convective flow behavior. The study concluded that CCV geometry with external cooling jackets gave better heat transfer, higher productivity, and simpler yeast removal. Although today it is believed the experimental results were inaccurate due to interruption of geometry, the work was a major conceptual step forward with respect to convective flow visualization in vertical fermenters.

By the mid-1970s, competitive pressures among the world’s larger commercial breweries drove the development of equipment that maximized the economy of scale. Strict control of CCV processes became the object of considerable industry focus in an effort to close a gap between theoretical and practical understanding of internal vessel fluid mechanics. Knudsen constructed small-scale glass and stainless steel models to observe convective flow patterns during brewing without the inherent disadvantages of the Delente study. Knudsen’s results secured earlier theories regarding the toroidal nature of internal flow patterns while remaining dimensionally consistent with commercial vessels.

In 1988, a strict engineering approach to CCV design was submitted by Larson and Brandon, and a preliminary description of turbulent behavior during cooling was described. In addition, the effects of tank geometry, beer temperature, cooling jacket temperature, and cooling jacket positioning were summarized. Time/temperature data were collected by adjustable immersion thermoprobes placed at several locations throughout the tank, and boundary layer data was collected at the wall via syringe. It was proposed by the authors that the best location for an average temperature measurement was at a fixed distance above the junction of the cone and cylindrical parts of the vessel. A follow-up to this study was presented in 1995 by Reuther, Brandon, Raasch, and Raabe. In this work, a 1/10th scale model was constructed to simulate the flow behavior of an existing commercial installation operating in the United States. The experimental and commercial cooling data obtained were in excellent agreement, thus establishing simulation as a design criterion for tank construction.

Recently, the first published attempt at non-intrusive, computational modeling of internal flow during CCV cooling was presented by Ishiguro, Mizutani, and Kuwahara. In their work, the energy, mass, and momentum conservation laws were solved using a multi-directional finite difference method and a supercomputer. Experimental temperature/time data were collected from a test tank at a brewery in Japan and showed excellent correlation with predicted results. In addition, several alternative designs to enhance the convective mixing in CCVs were tested.

The objective of this work is to advance theoretical understanding of flow in CCVs by presenting a computational scheme for solving an otherwise intractable beer fluid flow problem. The results will be validated by comparison with accepted analytical and field data collected by Brandon, et. al. The finished model will reproducibly simulate a CCV cooling operation at a representative geometry and provide information on the mixing aspects of convective flow, and a sampling point for the fluid mean temperature.

**FIELD PROCESS DESCRIPTION**

CCVs are vertically oriented, cylindrical vessels with a dished head at the top and a conical section fused to the bottom that reduces the cylinder diameter to a small orifice. Commercial equipment is normally specified by the ratio of the straight side cylinder height to the diameter, or aspect ratio. A two-dimensional tank schematic with an aspect ratio of unity is shown in Figure 1.

![CCV Aspect Ratio Definition](image)

In a normal lager CCV operation, wort produced in the brewhouse is pitched with yeast and fermented. The beer is then chilled from 15 ± 3 °C to a final temperature of 0 ± 2 °C by an applied cooling surface (exact temperatures are left to the brewer). During this time, buoyancy forces and turbulence will develop due to strong thermal gradients across the vessel wall and within the fluid, resulting in a slowly recirculating flow in the vessel. Yeast will become dormant and will not significantly alter the circulation pattern. The convective nature of the mixing is complicated by a flow inversion during the operation (~4 °C) where the density no longer increases with decreasing temperature (Figure 2).
After the inversion, the initial recirculation pattern will stagnate, then slowly reverse direction for the remaining portion of the process cycle, significantly lengthening the cooling process.

Several additional constraints challenge efficient processing of beer during cooling: (a) tank flow aids are avoided, as fittings, welds, or bracing may host contaminants, and (b) impeller agitation may sweep settled biomass into the beer, affecting its clarity, filterability, and sensory character. For these reasons, CCVs are generally unagitated tanks with smooth internal contact surfaces. Cooling is provided by heat transfer jackets constructed around the external shell of the tank, and generally occurs in a one- or two-step event called “crash cooling”. If a first crash cool is completed, the warm fermented beer is chilled to \( \sim 6 \, ^\circ C \). During a second crash cool, the temperature is further reduced to \( 0 \pm 2 \, ^\circ C \). It is also acceptable to perform both crash cools in one step. In this work, a second crash cool sequence is analyzed.

**TECHNICAL APPROACH**

An engineer’s research cache contains two main tools for testing new, existing, or retrofit designs: computational modeling and experimental modeling. In today’s rapid innovation environment, use of computational models provides advantages over experimental scale modeling in terms of cost and time required for data collection. Parameters used in simulation efforts may be adjusted for design optimization. Single variable or multivariate relationships can be studied quickly with a minimum setup time and within a specified error tolerance. Overall system properties are estimated by integrating solution values from all of the nodes. Since FEA is an iterative process, estimates systematically improve until even a massive number of nodes are under calculational control and within a specified error tolerance. The key step in FEA is the proper assembly of a linear algebraic system to solve the conservation equations at each node:

\[
\begin{align*}
\mathbf{\nabla} \cdot \mathbf{\rho \nu} &= 0 \\
\mathbf{\rho} \left( \frac{D\mathbf{v}}{Dt} \right) &= -\nabla p + \mu \nabla^2 \mathbf{v} + \rho g \\
\mathbf{\rho} \mathbf{C_p} \left( \frac{DT}{Dt} \right) &= \kappa \nabla^2 T + \mu \Phi
\end{align*}
\]

Equation 1 symbolically states that all mass changes must counteract and balance (thus equalling zero), and are a function of fluid density \( \rho \) and velocity \( \mathbf{v} \) in all dimensions (symbolized by \( \nabla \)). Equation 2 defines the momentum changes \( (D\mathbf{v}/Dt) \) in the fluid as functions of pressure \( p \), viscous \( \mu \), and gravitational forces \( g \). Equation 3 describes the transient thermal behavior of the system \( (DT/Dt) \) in terms of fluid heat capacity \( C_p \), conductivity \( k \), temperature \( T \), and frictional heating \( \mu \Phi \). Proper application of the equations requires accurate fluid data, equipment dimensions, and a cursory physical understanding of the system. The strength of FEA is a capability to rapidly evaluate systems of differential equations by substituting simpler algebraic expressions. The basic assumption of FEA is that a fluid behaves as a single entity, or continuum. This continuum, with an infinite number of solutions, can be broken down into a finite set of small, representative subregions called elements. When assembled, these elements form a mesh, or grid, as shown in Figure 3 on next page.

Each fluid element may be re-sized until all points inside are behaviorally indistinguishable. This is known as local equilibrium. Then a representative point, called a node, is chosen and the conservation laws are solved for the point by FEA. For reasons of mathematical efficiency, nodes are typically selected at element intersections. Overall system properties are estimated by integrating solution values from all of the nodes. Since FEA is an iterative process, estimates systematically improve until even a massive number of nodes are under calculational control and within a specified error tolerance. The key step in FEA is the proper assembly of a linear algebraic system to solve the conservation equations at each node: 

\[
\begin{align*}
\mathbf{\nabla} \cdot \mathbf{\rho \nu} &= 0 \\
\mathbf{\rho} \left( \frac{D\mathbf{v}}{Dt} \right) &= -\nabla p + \mu \nabla^2 \mathbf{v} + \rho g \\
\mathbf{\rho} \mathbf{C_p} \left( \frac{DT}{Dt} \right) &= \kappa \nabla^2 T + \mu \Phi
\end{align*}
\]
The concept of the preceding equation is not abstract. In brief, the total momentum and energy forces are a sum of:

- Density and heat energy variations due to temperature and velocity changes
- Convective, pressure, and viscous force variations due to temperature and velocity changes

Equation 4 works by first accepting pre-calculated initial guesses for unknown properties at each node. Using algebra, an attempt is made to balance unknown momentum and energy forces at the nodes to a specified error tolerance. Once satisfied, the solution values are used as new initial guesses, and so on, until the conservation laws for the entire system of nodes agree. This condition is known as convergence. The sections that follow derive the critical unknowns for CCV solution, and the proper method for handling these factors computationally.

\[
\begin{bmatrix}
\rho & 0 \\
0 & C_p
\end{bmatrix}
+ \begin{bmatrix}
\rho v \Phi, \rho g \\
\rho T, kT
\end{bmatrix}
= \begin{bmatrix}
momentum force \\
energy force
\end{bmatrix}
\]

**EQUATION 4**
Linear Algebra Statement of the Conservation Laws.

**DIMENSIONLESS ANALYSIS**

Free convective systems exhibit unique momentum and heat transfer characteristics. Thus, it is possible to simplify parts of the conservation laws to make the equations shorter, easier, and faster to solve. A technique known as the Buckingham-\(\pi\) method\(^{10}\) can be performed to better understand how the conservation equations are properly reduced for FEA. From the conservation Equations 1-3, a free convective system can be defined theoretically by a set of physical parameters, called primitives:

\[
\text{System} = f(l, g, v, k, \rho, C_p, T, p)
\]

**EQUATION 5**
Primitive System Definition

These variables are termed primitives because the quantities are the simplest measurements that uniquely describe the system. Of the nine preceding terms, all exist within the four fundamental dimensions of mass, length, time, and temperature. Using the Buckingham-\(\pi\) rule, convection is therefore described by \((9 \text{ primitives} - 4 \text{ dimensions}) = 5\) dimensionless factors known as \(\pi\) terms that "could" describe the bulk system. The objective is to envelop all primitive variables in at least one dimensionless quantity, thus relating all quantifiable physical changes in the system. The result of the analysis is presented in Table 1.

The number of dimensionless quantities is reduced by considering how convective systems actually operate. During CCV cooling, there is relatively little pressure change in the vessel, thus the Euler number may be disregarded. Since fluid velocities are low in free convection, the Reynolds and Brinkmann numbers are not critical for system definition. Only the Grashof and Prandtl numbers remain as essential system descriptors. In combination, the Prandtl and Grashof numbers form a single product known as the Rayleigh number:

\[
N_{Ra} = N_{Pr} \cdot N_{Gr} = \frac{C_p \rho^3 g l^3}{\mu k}
\]

**EQUATION 6**
Rayleigh Number Definition

Thus, a free convective system is characterized by only one dimensionless quantity that dictates all the necessary primitives to define the fluid behavior: (a) heat capacity, (b) density, (c) gravity, (d) length, (e) viscosity, and (f) thermal conductivity. Temperature is implicitly contained in the density expression.

**DERIVATION OF TURBULENT TRANSFER QUANTITIES**

As shown in the preceding section, dimensionless analysis is an excellent method for deriving critical variables needed to describe the bulk properties of a system. However, these quantities do not provide a detailed description of turbulent behavior exhibited by beer during cooling in CCVs. In a turbulent flow domain, three distinct regions are recognized: (a) a laminar sub-layer, or non-turbulent film near a boundary, (b) a transitional...
region where fluid parcels are randomly exiting and re-entering a turbulent stream and, (c) a turbulent region, where the random and chaotic momentum and energy fluctuations are fully developed. It is important to note that energy, mass, and momentum is conserved, even in turbulent flow. A simple diagram of a turbulent boundary layer is included in Figure 4.

When describing turbulent convective heat transfer with the Rayleigh number, it is accepted that values of $N_{Ra} < 1.7 \times 10^3$ are purely conductive in nature, $N_{Ra} > 1.7 \times 10^3$ indicate at least weak convective forces are present in the system, and values of $N_{Ra} > 1.0 \times 10^9$ indicate that convection is turbulent with little or no conduction. Values of $N_{Ra}$ between $10^5-10^9$ are considered transitional. To determine the extent of turbulence, the Rayleigh numbers at temperature points during a beer cooling cycle were calculated assuming a wall temperature of 2°C. Standard lager primitive values were used. The result is shown in Figure 5.

From Figure 5, it is clear that a turbulence model is needed to characterize the cooling behavior. This model must be flexible enough to be switched on during the early turbulent portion of the cooling cycle, off as needed while the flow is transitional, and back on again toward the latter of the cooling cycle. To accomplish this end, a general purpose turbulence solution method known as the $k-\varepsilon$ model was used. The method was collectively derived by engineers, mathematicians, and physicists during the early to mid-1900s, and has gained much popularity for an ability to solve a broad spectrum of turbulent flows. In basic form, the model is represented by the following:

### TABLE 1
Dimensionless Quantities Describing Free Convection

<table>
<thead>
<tr>
<th>Equation derivation</th>
<th>Solution</th>
<th>Dimensionless Quantity</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\pi_1 = (\rho)^a(g)^b(f)^c(\mu)^d$</td>
<td>a: 2</td>
<td>$N_{Gr} = \frac{\rho^2 g l^3}{\mu^2}$</td>
<td>$\frac{\text{Buoyancy forces}}{\text{Viscous forces}}$</td>
</tr>
<tr>
<td></td>
<td>b: 1</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>c: 3</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>d: -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\pi_2 = (C_p)^a(k)^b(\mu)^c$</td>
<td>a: 1</td>
<td>$N_{Pr} = \frac{C_p \mu}{k}$</td>
<td>$\frac{\text{Momentum diffusion}}{\text{Thermal diffusion}}$</td>
</tr>
<tr>
<td></td>
<td>b: -1</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>c: 1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\pi_3 = (\mu)^a(v)^b(k)^c(T)^d$</td>
<td>a: 1</td>
<td>$N_{Br} = \frac{\mu v^2}{kT}$</td>
<td>$\frac{\text{Viscous dissipation}}{\text{Conductive dissipation}}$</td>
</tr>
<tr>
<td></td>
<td>b: 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>c: -1</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>d: -1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\pi_4 = (\mu)^a(v)^b(l)^c(\rho)^d$</td>
<td>a: -1</td>
<td>$N_{Re} = \frac{v \rho}{\mu}$</td>
<td>$\frac{\text{Inertial forces}}{\text{Viscous forces}}$</td>
</tr>
<tr>
<td></td>
<td>b: 1</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>c: 1</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>d: 1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\pi_5 = (\rho)^a(\rho)^b(v)^c$</td>
<td>a: 1</td>
<td>$N_{Eu} = \frac{P}{v^2 \rho}$</td>
<td>$\frac{\text{Pressure forces}}{\text{Inertial forces}}$</td>
</tr>
<tr>
<td></td>
<td>b: -1</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>c: -2</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
\[ \mu_t = \mu \pm \rho C_{\mu} \left( \frac{k^2}{\varepsilon} \right) \]

**EQUATION 7**

**K-\varepsilon** Turbulent Viscosity Model

Since both \( \kappa \) and \( \varepsilon \) depend upon an *in situ* velocity and density, two additional terms are added to the system of energy, mass, and momentum conservation equations. The principal advantage of the method, despite the obvious expense computationally, is a sensitivity to energy fluctuation and fluid environment, which more closely represents real behavior.

**MODELING OF TURBULENT HEAT TRANSFER**

In addition to momentum, turbulent \( \kappa-\varepsilon \) terms are required for the thermal conductivity and are by definition

\[ k_t = k \pm \frac{C_{\mu} \left( \rho_0 C_{\mu} \left( \frac{k^2}{\varepsilon} \right) \right)}{\sigma_t} \]

**EQUATION 8**

**K-\varepsilon** Turbulent Conductivity Model

**MODELING OF BOUYANCY BEHAVIOR**

With the definition of turbulent viscosity and conductivity terms complete, the next step is to locate a working density model for the beer. During CCV cooling, differences in density are a function of temperature only (since the vessel pressure is static). This behavior is characteristic of an incompressible fluid. In such a fluid, a suitable representation of temperature induced density fluctuation is known as the Boussinesq approximation represented as:

\[ (\rho - \rho_0)g = -\rho_0 [\beta_T (T - T_0)]g \]

**EQUATION 9**

**Boussinesq Fluid Density Model Definition**

The coefficient of thermal expansion \( \beta_T \) is a functional relationship between density and temperature at constant pressure:

\[ \beta_T = \left. -\frac{1}{\rho} \left( \frac{\partial \rho}{\partial T} \right) \right|_p \]

**EQUATION 10**

**Thermal Expansion Coefficient Definition**

**STATIC PRESSURE MODELING**

By a strict fluid mechanical definition, density changes at a fixed volume generate a pressure force. In low pressure convection, these forces tend to be trivial, and FEA practitioners have developed the *penalty function method* to represent small pressure changes that occur in Boussinesq fluids. The penalty function approximates pressure as follows:
In low velocity systems where the density changes are small, an analogy can be drawn between an incompressible fluid and an incompressible elastic solid, making the penalty function, \( \lambda \), analogous to a volume stress, or bulk modulus of elasticity.\(^{115} \) Fluid motion velocities for CCVs are on the order of \( 10^{-3} \) m/s, while pressure gradients within Boussinesq fluids can be six to nine orders of magnitude smaller. Similar to the result of dividing large numbers by small ones, the solution of large algebraic matrix systems with trace pressure values would be mathematically unstable. By selecting a representative velocity, position, and constant, pressure terms can be safely substituted by quantities of larger magnitude. This increases computational efficiency during iteration with no appreciable loss in accuracy.

**FORMULATION OF THE CONVECTIVE CONSERVATION EQUATIONS**

Collection of the buoyancy, pressure, turbulent conductivity, and turbulent viscosity models gives the following modified conservation laws in all relevant dimensions:

\[
\rho \left( \frac{Dv}{Dt} \right) = \lambda \nabla v + \mu \nabla^2 v - \rho_0 \beta_T (T - T_0) g
\]

**EQUATION 12**

Conservation of Turbulent, Buoyant Momentum

\[
\rho C_p \left( \frac{DT}{Dt} \right) = k_f \nabla^2 T + \mu \Phi
\]

**EQUATION 13**

Conservation of Turbulent, Convective Energy

The derivation of the conservation laws for solution by FEA is complete. The only task remaining is collection of primitive physical data for input into the system of elements. Recall, the Rayleigh number dictates the critical variable data, and accepted values for beer primitives are presented in Table 2.

Lager beer property data was obtained from the Siebel Institute of Technology in Chicago, IL, USA.\(^{116} \) The values of \( \beta_T \) were derived from the density/temperature profile of lager beer using data from the Institute. A first derivative of density with respect to temperature was determined, then multiplied by the inverse of the instantaneous density to give the coefficient of thermal expansion, \( \beta_T \). Since the physical properties of beer closely resemble water, the heat capacity, thermal conductivity, and viscosity are assumed constant over the temperature range of interest.

**SOFTWARE**

The computer program FIDAP was employed to perform FEA calculations for Equations 12 and 13. FIDAP is a commercially available software program from Fluent, Incorporated, of Lebanon, NH, USA. The code, comprised of over three million lines, is written entirely in FORTRAN. Contained within FIDAP are several modules that are used for computer aided design (CAD) of the vessel, physical data entry and solution method selection, algebraic matrix assembly, iterative solution, results processing, and flow drawing generation. The selection of FIDAP as the code of choice was based upon capability, training, technical support, reasonable startup cost, and availability on the Microsoft Windows NT platform. In addition, FIDAP had a particular advantage in solving turbulent, convective systems.

**EXPERIMENTAL PROCEDURE**

Before CAD and meshing of a simulated tank with FIDAP, engineering and dimensional information was obtained from the literature and communications with the tank manufacturer.\(^{17,8,117} \) Specifications taken from the engineering drawings for both the commercial and experimental tanks are presented in Table 3. The large tank dimensions were provided by the manufacturer of an existing commercial installation in La Crosse,
TABLE 3

<table>
<thead>
<tr>
<th>Tank</th>
<th>Height</th>
<th>Diameter</th>
<th>Cone Angle</th>
<th>Aspect Ratio</th>
<th>Volume</th>
</tr>
</thead>
<tbody>
<tr>
<td>Commercial tank</td>
<td>4.87 m</td>
<td>4.64 m</td>
<td>45° to horizontal</td>
<td>1.05</td>
<td>100.1 m³</td>
</tr>
<tr>
<td>Experimental tank</td>
<td>0.914 m</td>
<td>0.914 m</td>
<td>45° to horizontal</td>
<td>1.00</td>
<td>0.1001 m³</td>
</tr>
<tr>
<td>FEA tank</td>
<td>0.914 m</td>
<td>0.914 m</td>
<td>45° to horizontal</td>
<td>1.00</td>
<td>0.1001 m³</td>
</tr>
</tbody>
</table>

WI, USA. All experimental data from the installation were provided by brewery personnel. The experimental tank was prepared by the same manufacturer and delivered to Washington University in Saint Louis for on-site testing. All data was provided by the experimenter. The small tank was specifically designed to be dimensionally consistent with the commercial fermenter.

Since turbulence is exhibited during cooling, grading the mesh toward walls, phase boundaries, and suspected regions of chaotic flow was necessary to characterize the beer behavior. Once a suitable mesh was developed, processing conditions (known as initial and boundary conditions), fluid data, and the κ-ε turbulence model were imposed on the geometry. With FIDAP, this was completed through a procedure termed “entity definition”, whereby a set of system or fluid characteristics were attached to parts of the CAD drawing. For example, the flow field was defined as a “fluid” entity to which all of the distinguishing properties of beer were attached. Another type was a “wall” entity, which was used to assign temperatures and turbulence models to boundaries. Thus, entities were parts of the system upon which the fluid and boundary conditions acted and interacted. CCY entities are shown in Figure 6.

After the entities were defined, the ensemble was loaded into FIDAP and the solver was run. In order to conserve computer memory, calculations were completed in an axi-symmetric fashion, based upon the assumption that each two-dimensional half of a CCY was a mirror image. If axial rotational flow (which would comprise a third angular dimension) were a significant factor, a reasonable solution would not be generated by the solver.

To simulate a real operation, the “cooling jackets” (upper wall) were activated at \( t = 0 \), causing a sharp thermal gradient across the wall. The cone (lower wall) remained ambient. As the beer (fluid) was chilled, the temperature of the wall remained constant, while the beer, cone, and upper phase boundary (top) were free to attain any velocity or temperature consistent with convergence conditions. As thermal gradients became less pronounced, the turbulent contribution to the fluid flow diminished until transitional flow was present in the vessel. The turbulence models were automatically shut off, and the solution proceeded accordingly.

**MESH REFINEMENT**

When characterizing turbulent flow, a significant consideration is how and where to position the first row of nodes away from the wall. The objective is to set the edge of the mesh within the turbulent region, but not inside the laminar sublayer. The reasoning behind this restriction is that κ-ε models are not adept at resolving laminar flow. Such a situation results in unproductive over-iteration. If nodes are properly placed in the boundary layer, an additional near-wall model will effectively resolve the laminar region from the wall to the first row of nodes. The challenge is determining where the two models meet in the fluid domain. Computational practitioners have developed a measuring tool to simplify node placement efforts, as:

\[
y^* = \frac{y \sqrt{\rho \tau}}{\mu}
\]

**EQUATION 14**

Dimensionless Boundary Layer Thickness
Referring to Figure 4, values of $y^+ < 15-30$, indicate the first row of nodes is near the edge of the laminar sublayer. Values between 30 and 100 empirically indicate the mesh is appropriately established in the turbulent stream. Mesh adjustment will affect the $y^+$ calculation, giving the experimenter direct response for refinement. Laminar flows are resolved by assuming a local and physical data proceed through “virtual” time. Movement smooth computational assumption at the wall, and (h) the wall contains the conservation equations, unknown variables, face variance of less than $0.13 \mu m$ between a peak or valley from the mean surface height. The significance of this grade of finish is twofold: (a) the small surface allows for an ideally smooth computational assumption at the wall, and (b) the wall surface will not retain yeast cells or beer/wort spoilers, which have typical particle diameters of $> 35 \mu m$ and $> 0.45 \mu m$, respectively. Thus, no special accommodation for surface roughness was made during solution since the material surface characteristics did not interfere with the laminar or turbulent streams.

**TIME STEPPING ALGORITHM**

In transient systems, the matrix assembly (Equation 4) that contains the conservation equations, unknown variables, and physical data proceed through “virtual” time. Movement through this dimension requires a means of resolving time-based rates of change. Assume the quantity $M$ represents the algebraic matrix of conservation laws. Since time rates of change can be represented by differentials, let the definition of the time derivative be:

$$M'(t) = \lim_{\Delta t \to 0} \frac{M(t + \Delta t) + M(t)}{\Delta t}$$

**EQUATION 15**

**Definition of the Derivative**

Let the change in time ($\Delta t$) be sufficiently small, such that:

$$M'(t) \equiv \frac{M(t + \Delta t) + M(t)}{\Delta t}$$

**EQUATION 16**

**Derivative Simplification**

Solving for the quantity $M(t + \Delta t)$ will give an estimate of changes in $M$ at the next time step ($t + \Delta t$). The above derivation is known as Euler’s Method, which rapidly computes time derivatives. FIDAP manages the accuracy and expediency of Euler’s Method by adjusting time step sizes as error tolerances allow. Relaxation, a form of prior result carryover, is also applied to the system to stabilize the calculations. When a system is relaxed, a fraction of preceding solutions is mixed with a fraction of subsequent solutions, avoiding spurious over and undershooting of derivatives during iteration.

**RESULTS AND DISCUSSION**

**REFINEMENT OF THE COMPUTATIONAL MODEL**

Demonstrating grid resolution and reproducibility of the mesh is the single best way to establish credibility for an FEA model. A FEA system is grid resolved when: (a) the algebraic system converges reproducibly to the same solution, despite further diffusion or concentration of the mesh, and (b) increasing the total number of nodes gives no further solution accuracy. Output must also agree with experimental data collected from a real system of the same design. Post-processing and visual inspection of the flow fields should be performed to ensure that the FEA results are not coincidental artifacts.

While testing for grid convergence, the objective was to maximize calculation stability and minimize computation time while using the coarsest mesh possible. For the cylindrical portion of the tank, mesh grading was completed by bisecting the axisymmetric drawing with respect to both length and width, resulting in two secondary centerlines for each section. Referring to Figure 3, a 5:1 gradient ratio was directed axially from the tank secondary centerline to both the top and bottom of the domain. Similarly, a 4:1 gradient ratio from a secondary centerline to both the wall and the line of axial symmetry was completed radially. For the lower conical portion, both the radial and axial directions were graded in a 4:1 ratio from secondary centerlines. After the required number of nodes was situated with adequate grading, FEA proceeded smoothly and reproducibly at $y^+$ values between 2-15. The dimensionless boundary layer values are summarized as a function of time in Figure 7.

**FIGURE 7**

**Predicted $y^+$/time Data for the CCV Cooling Cycle**
The low predicted value for $y^+$ indicated the turbulent boundary layer in the CCV was exceptionally thin. It was observed that a mesh constructed of ~1200 nodes was required to converge reproducibly using FIDAP. The ratio of axial to radial nodes was approximately 2:1.

**TIME NORMALIZATION**

The real purpose of FEA is to describe the mechanism of a large system by substituting a virtual one. A problem encountered when modeling real systems is how to compare chronological data at several length scales. The best way to examine data of this nature is through a time normalization process. Specifically, the conversion of time based quantities to a dimensionless form eliminates the impact of vessel proportions upon the data. This results in the engineer being able to directly overlay and compare experimental results visually. Brandon’s [81 1/10th scale CCV results proved this comparison held for any two geometrically consistent systems. For example, the length scale of the commercial tank was ten times that of the model, and the commercial process took ten times as long to reach the same temperature objective. A direct comparison of Brandon’s results with the FEA output can be made by converting temperature/time data to temperature/dimensionless time quantities (as a % of the total cooling cycle time), according to the following conversion:

$$\text{Dimensionless time (\%) = \frac{\text{Elapsed time}}{\text{Total time}}}$$

**EQUATION 17**

Time Normalization

Figure 8 compares the large and small scale experimental data of Brandon [81 with the results obtained from FEA. Brandon’s model tank dimensions were used for the simulation.

A departure analysis of the temperature data for all three tanks was completed, and comparison of the data sets gave the following conclusions: (a) the 1/10th scale data was within 4.0% of the commercial CCV over the entire cycle, (b) the FEA results obtained were within 2% of small scale CCV data, and (c) the FEA results obtained were within 3% of the commercial CCV results. Figure 8 reflects this agreement.

Representative streamline function and vector field results are presented in Figures 9 through 14. Streamlines are mathematical statements that represent fluid parcel pathways in the vessel, and are indicative of the flow pattern. Vector field plots compliment the streamline plots by indicating the magnitude and direction of the flow at points throughout the fermenter. The graphics were generated by the FIDAP post-processor, and visual inspection of the output resembles previous work by Ishiguro [91 and Knudsen [51. Particular attention is directed to streamline/vector plots in Figures 11, and 12, which demonstrate the ability of FIDAP to simulate a density inversion during the beer cooling cycle, capturing a characteristic multi-cell flow field.
TIME STEPPING METHODOLOGY

A plot of the time stepping scheme for solution is shown in Figure 15. It was relatively simple to bracket the proper time stepping sequence while using FIDAP. If the time steps were too coarse, the solver diverged quickly and produced error messages due to excessive truncation. When the time steps were too fine, roundoff errors predominated and the solver did not converge. When the time steps were within an acceptable range, the solver smoothly self-adjusted according to the error tolerance value and converged reproducibly.

The first time step taken was at 0.005 s, and the time step size was progressively increased until the turbulence diminished. When the system exhibited transitional flow, larger time steps were taken until an upper limit of 30 s per time step was reached.
Time steps > 40 s tended to cause a significant amount of truncation error. Reproducible divergence was obtained when fixed time step experiments were run, which demonstrated the mandatory use of flexible time stepping techniques.

**MEAN TEMPERATURE MEASUREMENT**

Figure 16 shows the result of a departure analysis of the integrated mean temperature for the entire tank versus the instantaneous temperature at each node. The highlighted points in the figure represent a positive or negative departure from the mean no greater than 0.3°C, and are the largest single cluster of nodes detected by FEA exhibiting this property. Although no “perfect” point to measure the mean temperature exists, the indicated region is the closest possible location to an ideal. With the exception of the wall boundary layer where thermal gradients are pronounced, the majority of the tank is within 0.5 to 1.0°C of the mean temperature (at this aspect ratio) for most of the cooling cycle.

**FIGURE 16**

Nodes with lowest Departure from the Mean Temperature

From Brandon[8], it is suggested that a temperature probe be located at 20-25% of the distance of the tank straight side, originating at the junction of the cone and the cylinder. FEA shows that positioning a thermoprobe in this fashion yields a mean departure of ~0.32°C for the entire unit operation, including the stagnant inversion event. The significance of this conclusion is that previous experimentally based recommendations regarding single point temperature measurement, hold for this geometry, and are validated by FEA. In addition, measurement within this volume of consistent thermal and velocity behavior, diminishes errors in probe placement and will not adversely affect the mean temperature reading.

**FLUID VELOCITY DURING CCV COOLING**

Fluid velocities are relatively low in free convection, and a CCV is no exception. At ~4°C, the onset of transitional flow slows the velocity field dramatically. This indicates why cooling beer through a density inversion is problematic to brewers and engineers. The onset of inversion and change to transitional flow occur simultaneously, without any recourse to mechanically mix the stagnating fluid. Referring to Figure 17, the turbulent portion of the cooling cycle is complete within the first 20% of the total time necessary to complete the operation. Any notable mixing beyond this point is achieved by conduction, density inversion, and weak buoyancy effects.

![Figure 17: Velocity/Time Data for the Uni-Tank Cooling Cycle](image)

**FIGURE 17**

Velocity/Time Data for the Uni-Tank Cooling Cycle

**DESIGN IMPLICATIONS**

Assuming a standard CCV construction (no baffles, fillets, or agitation), FEA provides a few general statements regarding tank design:

- The Rayleigh number describes the bulk thermal behavior of dimensionally consistent scaled CCV systems.
- During cooling, lager beer will exhibit both turbulent and transitional flow behaviors regardless of vessel size.
- Before density inversion, the best location for a primary cooling jacket is near the junction of the cone and cylinder where the internal fluid recirculation is most prevalent. A good secondary location is near the headspace of the cylinder to redirect the flow toward the cone. During this time, a strong recirculation zone forms near the cone/cylinder junction.
- During density inversion, the ideal location for a primary cooling jacket is along the entire cylinder length, as conduction dominates buoyant mixing. Double cell flow will be observed during this event.
- After density inversion, the ideal location for a primary cooling jacket is near the upper fluid level where the internal fluid recirculation is most prevalent. A good secondary location is near the junction of the cone and cylinder to redirect the flow toward the headspace. Relatively little fluid motion occurs after the inversion due to the small thermal gradient across the wall into the beer.
CONCLUSION

Finite element analysis of a CCV cooling cycle was performed and compared with field data presented in the literature. The quantitative temperature/time and qualitative now results were in agreement with experiment and provided validation for \(k-\varepsilon\) turbulence modeling of transient and turbulent free convective flow. In addition, a single-point measurement position mentioned in the literature was substantiated numerically.

Approximation and stability theory used to arrive at an acceptable solution was presented for employment and guidance for new problems and modeling efforts.

NOMENCLATURE

\(C_p\) .... Fluid heat capacity  
\(C_{\mu}\) .... Turbulence two-equation model constant  
g ..... Gravitational acceleration  
k .... Fluid thermal conductivity  
k\(_t\) .... Fluid turbulent thermal conductivity  
l .... Characteristic Length  
\(M\) .... Matrix notation  
p ...... Pressure  
t ...... Time  
\(T\) .... Temperature  
v ...... Fluid velocity  
x ...... Arbitrary position or length (horizontal)  
y ...... Arbitrary position or length (vertical)  
\(x^+\) .... Dimensionless distance in the boundary layer  
\(\Delta t\) .... Change in time  
\(\beta\) .... Thermal expansion coefficient  
\(\epsilon\) .... Turbulent dissipation  
\(\Phi\) .... Fluid viscous dissipation  
\(\kappa\) .... Turbulent kinetic energy  
\(\lambda\) .... Penalty function constant or pressure dissipation  
\(\mu\) .... Fluid viscosity  
\(\mu_t\) .... Fluid turbulent viscosity  
\(\pi\) .... Buckingham-\(\pi\) term  
\(p\) ...... Fluid density  
\(\rho_0\) .... Reference fluid density  
\(\sigma_t\) .... Turbulence two-equation model constant  
\(\tau\) .... Turbulent shear stress  
\(f\) .... Buckingham-\(\pi\) function  
\(\nabla n\) .... Multidirectional derivative form for an arbitrary variable, \(n\)  
\(\partial n\) First derivative form for an arbitrary variable, \(n\)

ACKNOWLEDGEMENTS

The authors wish to express their appreciation for Mr. Jim Larson, Siebel Institute of Technology, Chicago, IL, and Mr. Chad Kelly, Enerfab Incorporated, Cincinnati, OH, for their excellent advice and background information, project guidance, and provision of scientific data.

REFERENCES

ADDENDUM

FIDAP SOFTWARE

FIDAP is a commercially available fluid flow modeling and simulation program that is designed and tested to run on all existing computer operating systems. The package name, an acronym for Fluid Dynamics Analysis Package, is available from Fluent, Incorporated, of Lebanon, NH (USA). The FEA code, comprised of over three million lines, is written entirely in FORTRAN, and provides a sophisticated, reliable, and simple implementation of FEA for the end user. Contained within FIDAP are several smaller modules that are used for: (a) computer aided design (CAD) of the physical structure, (b) entering dimensional data, physical properties, and a solution method, (c) automatically assembling an algebraic matrix from simplified partial differential equations (PDE), (d) generating an iterative solution for the matrix, and (e) post-processing the results to generate finished drawings. The capabilities of FIDAP are far ranging, rendering it particularly useful for the research environment as well as industry. The program has been applied successfully in the areas of electronics, automotive design, metallurgy, HVAC, polymers, chemical kinetics, biomedicine, aerospace, mechanical engineering, heat exchange, and crystalization, to name a few.

FIDAP has a particular advantage in solving turbulent, convective systems from its strong combustion background. The selection of FIDAP as the code of choice is based upon product capabilities, provision of training and technical support, reasonable startup cost, and availability on the Microsoft Windows NT platform. Upon purchase of FIDAP, Fluent, Inc. provides offsite personal training (2-3 days) at a Fluent training center and ongoing engineering support to ensure the user understands the software. The learning workshop trains a new user through several worked examples, and by familiarizing them with the graphical user interface (GUI) and menu driven command structure.

In practice, an experiment is typically begun by gathering all known physical quantities of the system. Examples include any vessel dimensions, fluid properties, temperatures, pressures, heat transfer data, etc. The system will then be drawn using FIDAP in one or multiple dimensions based upon the level of accuracy required of the solution. Geometry can also be imported from an AutoCAD file. Once the dimensions are properly set, boundary data is “attached” to the CAD drawing by a series of simple GUI menu commands (simply select a boundary, such as a vessel wall, and declare a temperature along the wall). This procedure is repeated until all temperatures, pressures, viscosities, thermal conductivities, and other engineering properties relevant to the problem are affixed to a boundary or fluid. The user will then select a solution technique from a library of pre-programmed models that suit the nature of the system behavior. The model will “read in” the boundary data, convert the PDA statements to an algebraic matrix system (this is completed in memory), and attempt to satisfy the conservation laws throughout the entire domain. Fluent provides several example calculations and reference tutorials to simplify the model selection process. The tutorials and learning workshop will train the user as to which properties are required for a given model and which appropriate PDA solution techniques bring the best chance of success. After the solution has converged (a calculation report will be generated after iteration is completed), the user can process the numerical results into a graphical form that can be used to judge the quality of the model or solution technique. The data can also be integrated to yield average system properties.

Based upon the results, further refinements are made and the iterative process is repeated until the solution is stable and reproducible. When deriving a heat and momentum transport mechanism, as in this work, the solution is compared with experimental data to validate the theoretical model and to gain insight into the fundamental mechanisms of the system behavior.

COMPUTER REQUIREMENTS AND SOFTWARE EXPENDITURES

The minimum requirements to run FIDAP 7.62, the version used for this work, are summarized in the following table (for an IBM-compatible PC):

<table>
<thead>
<tr>
<th>Requirements and Expenditures for FIDAP</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>PC Component</strong></td>
</tr>
<tr>
<td>------------------</td>
</tr>
<tr>
<td>Personal Computer</td>
</tr>
<tr>
<td>Processor</td>
</tr>
<tr>
<td>CPU</td>
</tr>
<tr>
<td>RAM</td>
</tr>
<tr>
<td>File System</td>
</tr>
<tr>
<td>Hard Disk Space</td>
</tr>
<tr>
<td>Other Drives</td>
</tr>
<tr>
<td>Mouse</td>
</tr>
<tr>
<td>Graphics Driver</td>
</tr>
<tr>
<td>Monitor Resolution</td>
</tr>
<tr>
<td>Monitor Resolution</td>
</tr>
<tr>
<td>Network Card</td>
</tr>
<tr>
<td>Software</td>
</tr>
<tr>
<td>Software</td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

**Learning institutional costs are approximately 10% of the commercial price.**