SPH modeling of natural convection around a heated horizontal cylinder: A comparison with experiments

F. Aragón, J. E. V. Guzmán
Instituto de Ingeniería UNAM
Circuito Escolar S/N
Ciudad Universitaria, Coyoacán
04510 Ciudad de México, Mexico
Email: micme2003@yahoo.com.mx
Email: jguzmanv@iingen.unam.mx

C. E. Alvarado-Rodríguez
Dirección de Cátedras CONACYT
Av. Insurgentes Sur 1582,
Crédito Constructor, Benito Juárez
03940 Ciudad de México, Mexico
Email: carlos.alvarado@conacyt.mx

L. Di G. Sigalotti
Departamento de Ciencias Básicas
Universidad Autónoma Metropolitana
(UAM-A), Av. San Pablo 180
02200 Ciudad de México, Mexico
E.mail: leonardo.sigalotti@gmail.com

I. Carvajal-Mariscal
Instituto Politécnico Nacional (IPN)
ESIME UPALM, Av. IPN s/n
07738 Ciudad de México, Mexico
E-mail: icarvajal@ipn.mx

J. Klapp
Instituto Nacional de Investigaciones
Nucleares, (ININ)
Carretera México-Toluca km. 36.5
La Marquesa, 52750 Ocoyoacac
Estado de México, Mexico
E-mail: jaime.klapp@inin.gob.mx

A. R. Uribe-Ramírez
Departamento de Ingeniería Química
DCNyE, Universidad de Guanajuato
Noria Alta S/N, 36000 Guanajuato
Guanajuato, Mexico
E-mail: agustin@ugto.mx

An experimental and numerical smoothed particle hydrodynamics (SPH) analysis was performed for the convective flow arising from a horizontal, thin cylindrical heat source enclosed in a glycerin-filled, slender enclosure at low Rayleigh numbers (1.18 \( \leq \) Ra \( \leq \) 242). Both the experiments and the SPH calculations were performed for positive (0.1 \( \leq \) \( \Delta T \) \( \leq \) 10 K) and negative (\(-10 \leq \Delta T \leq -0.1 \) K) temperature differences between the source and the surrounding fluid. In all cases a pair of steady, counter-rotating vortices is formed, accompanied by a plume of vertically ascending flow just above the source for \( \Delta T \) > 0 and a vertically descending flow just below the source for \( \Delta T \) < 0. The maximum flow velocities always occur within the ascending/descending plumes. The SPH predictions are found to match the experimental observations acceptably well with root-mean-square errors in the velocity profiles of the order of \( \sim 10^{-5} \) m s\(^{-1}\). The fact that the SPH method is able to reveal the detailed features of the flow phenomenon demonstrates the correctness of the approach.

Keywords: Incompressible smoothed particle hydrodynamics; Natural convection; Heat transfer; Confined motion; Boussinesq approximation; Particle Image Velocimetry (PIV)

Nomenclature

- Ra Rayleigh number.
- Pr Prandtl number.
- H Height of test cell, m.
- L Length of test cell, m.
- W Width of test cell, m.
- D Diameter of cylindrical heat source, m.
- m Mass, kg.
- \( \rho \) Mass density, kg m\(^{-3}\).
- v Fluid velocity vector, m s\(^{-1}\).
- T Fluid temperature, K.
- p Pressure, Pa.
- F Body force, m s\(^{-2}\).

*Corresponding author.
Heat transfer by natural convection in an enclosed space has received increasing attention in the past years because of its importance in many engineering applications. In particular, natural convective heat transfer around a single horizontal cylinder and from horizontal tube arrays in cavities has direct applications in the design of heat exchangers, nuclear reactors, solar heaters, radiators, thermal storage systems as well as in the cooling of electronic components and the transport of oil through pipelines below the water surface or at the sea floor. These applications have motivated for more than 50 years a considerable body of research on their flow and heat transfer characteristics. In particular, recent experimental and numerical analyses of heat transfer and flow characteristics of the air-water cross flow in cambered ducts have been addressed to raise the efficiency of heat exchangers [1].

Square and rectangular heated cavities have been widely explored both experimentally and theoretically due to their simpler geometry. The main concerns of these studies were first directed to laminar and turbulent natural convection around heated, horizontal cylinders. At small Rayleigh numbers (Ra < 10^5), the heat transfer from a horizontal cylinder behaves like a line heat source [2, 3, 4, 5]. For larger Rayleigh numbers (i.e., 10^4 ≤ Ra ≤ 10^5), the flow forms a laminar boundary layer around the cylinder [6, 7, 8, 9, 10, 11, 12], while at even higher Rayleigh numbers, the motion of the fluid due to buoyancy is turbulent [13, 14, 15]. However, most of the research in this area has focused on laminar convective flows, which are the simplest and the most common cases in natural convection. Earlier numerical analyses by Kuehn [6], Farouk and Gücüri [7] and Saitoh et al. [10] have all shown that heat transfer is maximum at the bottom of the cylinder and decreases towards the top of the cylinder. This was experimentally assessed by Cesini et al. [5], who also observed that for Ra > 10^5, oscillations of the heat flux at the surface of the cylinder appeared. Moreover, Karim et al. [9] performed experiments to study the effects of horizontal confinement on heat transfer around a cylinder for 10^3 ≤ Ra ≤ 10^5, finding that the heat flux around the cylinder increases with decreasing distance between the cylinder and the enclosure wall. For similar experiments, Cesini et al. [5] found that the heat flux reaches a maximum for an optimal wall-to-wall distance of 2.9 times the diameter of the cylinder. In contrast, when the flow medium around the cylinder is vertically confined a complex combination of effects can be produced. For example, experiments conducted by Koizumi and Hosokawa [16] for vertical confinement of natural convection around a hot horizontal cylinder by both conductive and adiabatic flat ceilings has revealed chaotic and oscillatory movements of the air above the cylinder. Further experiments in water by replacing the flat ceiling with a free-surface boundary were performed by Atmane et al. [11].

They found that the primary effect of the vertical confinement was an increase of the heat flux on the upper part of the cylinder for given separation distances from the free water surface, which is related to the large-scale oscillation of the thermal plume.

A number of numerical analyses have been performed to study natural convection from horizontal cylinders enclosed in square and rectangular cavities, using spectral element [17, 18], finite-difference [19], finite-element [3, 5], and finite-volume [20, 21] methods. Articles dealing with Smoothed Particle Hydrodynamics (SPH) simulations of natural convection are not abundant in the literature. Extensions of SPH to fluid problems with heat transfer were first implemented by Cleary [22] and Cleary and Monaghan [23] under the Boussinesq approximation. Natural convection from a heated cylinder in a square enclosure was subsequently modeled by Moballa et al. [24] using an incompressible SPH approach. Their results were in good agreement with those from previous analyses based on conventional methods. SPH simulations with a non-Boussinesq formulation were also presented by Szewc et al. [25] to study natural convection under large density variations due to high temperature gradients. More recently, SPH simulations of natural convection in a horizontal concentric annulus and in square cavities were reported by Yang and Kong [26] and Garoosi and Shakibaenia [27], respectively. Further SPH calculations of natural convection heat transfer in a differentially heated cavity were reported by Garoosi and Shakibaenia [28]. On the other hand, both lid-driven cavity and buoyancy-driven cavity SPH simulations were carried out by Hopp-Hirschler et al. [29], while forced convection heat transfer around a horizontal cylinder was also investigated using a weakly compressible SPH (WCSPH) scheme by Nasiri et al. [30]. SPH studies of buoyancy-driven flow were recently presented by Hopp-Hirschler et al. [31], where the formation of viscous fingers is modeled in miscible and immiscible systems.
The thermal interaction between a cylindrical source and a rectangular enclosure was first investigated numerically by Ghaddar [17], who predicted flow patterns and heat transfer rates for air over a wide range of Rayleigh numbers. In a more recent work, De and Dalal [19] studied numerically the effects of varying the cavity aspect ratio on the flow pattern with the aid of finite-difference techniques. The present study will further extend the capabilities of the SPH method to simulate natural convection heat transfer problems. In particular, a weakly compressible SPH (WCSPH) formulation [32] based on the Boussinesq approximation is employed to reproduce experimental results of the natural convection flow around a horizontal cylinder of small diameter in a slender cavity of square cross-section L and width W = 0.05 L. The SPH method is a fully Lagrangian, mesh-free scheme for fluid-flow simulations, where fluid elements are represented by a set of discrete particles that characterize the flow attributes. It presents several advantages over traditional Eulerian techniques because it can exactly simulate pure advection and deal with complex geometries and boundaries. It must be remarked that most of the previous work with this kind of geometry has focused, to the best of our knowledge, on investigating the process with low viscosity fluids such as air and water [33, 34]. However, certain applications are driven by thermo-hydraulic phenomena that involve fluids with significant viscosity differences. For example, in the oil industry, the production of heavy oil relies on fluid-based thermal treatments to enhance the mobility of the highly viscous fluids [35, 36]. Clearly, the results and conclusions obtained in previous studies cannot be directly extrapolated in such cases. In particular, the present case may be representative of a single section of a heat exchanger used to reduce the viscosity of heavy crude oil in storage tanks.

The governing equations and the SPH methodology used are described in Section 2. The experimental rig and procedure are given in Section 3, while Section 4 describes the numerical model and the implementation of the boundary conditions. The validation of the numerical scheme along with the experimental and numerical results are presented in Section 5. The predictions obtained from the SPH simulations are found to be in good agreement with the experimental results. Finally, Section 6 contains the main conclusions.

2 Numerical methodology
2.1 Governing equations

The equations describing the fluid motion and heat transfer around a horizontal cylindrical heat source are the continuity, the Navier-Stokes, and the energy balance equations. In Lagrangian form these equations are:

\[
\frac{dT}{dt} = \frac{1}{\rho c_p} \nabla \cdot (k \nabla T),
\]

where \(d/dt\) is the material time derivative, \(\rho\) is the density, \(v\) is the fluid velocity vector, \(p\) is the pressure, \(v = \nabla / \rho\) is the kinematic viscosity coefficient and \(\eta\) is the dynamic viscosity of the fluid, \(T\) is the temperature, \(c_p\) is the specific heat at constant pressure, and \(k\) is the thermal conductivity. Since at low Ra-values heat generation due to viscous dissipation is negligible [27], only the thermal conduction term is retained in the energy balance equation (3). This is true because the total entropy generation due to thermal dissipation is by far much more intense than that due to viscous dissipation at \(Ra < 10^3\) [27]. In the Boussinesq approximation, the body force \(F\) on the right-hand side of Eq. (2) is given by [37]

\[
F = -\beta (T - T_r) g,
\]

where \(g\) is the gravitational acceleration, \(\beta\) is the coefficient of thermal expansion, and \(T_r\) is a reference temperature. In most applications this is the fluid temperature at room conditions. For natural convection flow in the Boussinesq approximation, the two relevant dimensionless numbers are: the Prandtl number

\[
Pr = \frac{v}{\alpha} = \frac{c_p \eta}{k},
\]

where \(\alpha = \kappa / (\rho c_p)\) is the thermal diffusivity, and the Rayleigh number

\[
Ra = \frac{g \beta D^3 \Delta T}{\alpha v},
\]

where \(D\) is a characteristic length scale and \(\Delta T\) is the leading temperature difference between the heat source \((T_s = T_{\text{heat}})\) and the surrounding fluid. In particular, the value of \(Ra\) quantifies the ratio of buoyancy over viscosity in the fluid for a given Prandtl number.

Equations (1)-(3) are closed by a pressure-density relation of the form [38]

\[
p = B \left[ \left( \frac{\rho}{\rho_r} \right)^\gamma - 1 \right],
\]

where \(B = c_0^2 \rho_r / \gamma\), \(\rho_r\) is a reference density, \(\gamma = 7\) for liquids and \(\gamma = 1.4\) for gases, and \(c_0\) is a numerical speed of sound which is chosen to be 10 times higher than the maximum fluid velocity in order to keep the density fluctuations below 1% and satisfy the incompressibility condition [39]. The reference density \(\rho_r\) in Eq. (7) is allowed to change with the temperature according to the following expression

\[
\rho_r = \frac{\rho}{1 + \beta (T_s - T_0)}
\]

where \(T_s\) and \(T_0\) are, respectively, the final and initial temperatures in a numerical timestep \(\Delta t\).
2.2 SPH formulation

Equations (1)-(3) are solved numerically using a variant of the open-source code DualSPHysics [30], which relies on SPH theory [39][41]. The temporal variation of the fluid density at the position of particle is calculated according to the SPH representation [39]

\[
\frac{d\rho_a}{dt} = \sum_{b=1}^{n} m_b \delta_{ab} \cdot \nabla_a W_{ab}, \tag{9}
\]

where \(v_{ab} = v_a - v_b\), \(W_{ab} = W(|x_{ab}|, h)\) is the kernel interpolation, \(x_{ab} = x_a - x_b\), \(h\) is the width of the kernel (or smoothing length), and the sum is over \(n\) neighbours around particle \(a\). This form is Galilean invariant and is particularly convenient in the presence of free surfaces or density discontinuities. For the momentum equation (2), we use the following representation

\[
\frac{dv_a}{dt} = -\sum_{b=1}^{n} m_b \left( \frac{p_a + p_b}{\rho_a \rho_b} \right) \nabla_a W_{ab} + 2v_a \sum_{b=1}^{n} m_b \frac{v_{ab}}{\rho_b |x_{ab}|^2 + \epsilon^2} + g, \tag{10}
\]

where \(\delta_{ab} = (\rho_a + \rho_b)/2\) and \(\epsilon^2 = 0.01 h^2\) is a small correction factor used to avoid singularities when \(|x_{ab}| \ll 1\). Following Cleary [22], the SPH form of the energy equation is given by

\[
\frac{dT_a}{dt} = \frac{2\kappa}{\rho_a c_p} \sum_{b=1}^{n} m_b T_{ab} x_{ab} \cdot \nabla_a W_{ab}, \tag{11}
\]

where \(T_{ab} = T_a - T_b\). This form ensures that the heat flux is automatically continuous across material interfaces. In its original form derived by Cleary [22] with \(\kappa \rightarrow 2\kappa_b \kappa_a / (\kappa_a + \kappa_b)\), it allows multiple materials with substantially different conductivities and specific heats to be accurately modelled. In a Lagrangian frame of reference, the SPH particles are moved according to

\[
\frac{dx_a}{dt} = v_a, \tag{12}
\]

which must be solved simultaneously with Eqs. (9)-(11).

Since direct evaluation of second-order derivatives of the kernel function is not required, a low-order Wendland \(C^2\) function is used as the interpolation kernel [42]

\[
W(q, h) = \frac{7}{h^6} (1 - q)^4 (1 + 4q), \tag{13}
\]

for \(0 \leq q < 1\) and zero otherwise, where \(q = |x - x'|/h\). Wendland functions have positive Fourier transforms and so they can support arbitrarily large numbers of neighbours without suffering from a close pairing of particles [42][43][44]. In addition, Wendland functions are very reluctant to allow for particle motion on a sub-resolution scale, and in contrast to most commonly used kernels they maintain a very regular particle distribution, even in highly dynamical tests as occurs in turbulent regimes [45].

The time integration of Eqs. (9)-(12) is performed using the Verlet algorithm provided by DualSPHysics, where the density, velocity, position, and temperature of particle \(a\) are advanced from time \(t^n\) to \(t^{n+1} = t^n + \Delta t\) using the following steps

\[
\rho_a^{n+1} = \rho_a^n + 2\Delta t \left( \frac{d\rho_a}{dt} \right)^n, \quad v_a^{n+1} = v_a^n + 2\Delta t \left( \frac{dv_a}{dt} \right)^n, \quad x_a^{n+1} = x_a^n + \Delta t v_a^n + 0.5 \Delta t^2 \left( \frac{dx_a}{dt} \right)^n, \quad T_a^{n+1} = T_a^n + \Delta t \left( \frac{dT_a}{dt} \right)^n. \quad (14)
\]

Numerical coupling of the discrete SPH equations is ensured during the evolution by alternating the above steps by the form

\[
\rho_a^n = \rho_a^n + \Delta t \left( \frac{d\rho_a}{dt} \right)^n, \quad v_a^n = v_a^n + \Delta t \left( \frac{dv_a}{dt} \right)^n, \quad x_a^n = x_a^n + \Delta t v_a^n + 0.5 \Delta t^2 \left( \frac{dx_a}{dt} \right)^n, \quad T_a^n = T_a^n + \Delta t \left( \frac{dT_a}{dt} \right)^n. \quad (15)
\]

The time step \(\Delta t = t^{n+1} - t^n\) to ensure stability of the above scheme is calculated as the minimum between the following timesteps:

\[
\Delta t_f = \min_a \left( \frac{h |dv_a|/dt|^{-1}}{1} \right)^{1/2}, \quad \Delta t_{cv,a} = \max_b \left[ h |x_{ab} - v_{ab}| / |x_{ab} \cdot x_{ab} + \epsilon^2| \right], \quad \Delta t_{cv} = \min_a \left[ h (c_a + \Delta t_{cv,a})^{-1} \right], \quad \Delta t_c = \min \left( 0.1 \rho_a c_p h^2 / \kappa \right), \quad \Delta t = \min (\Delta t_f, \Delta t_{cv}, \Delta t_c), \quad (16)
\]

where \(c_a\) is the speed of sound of the fluid.

2.3 Numerical model and boundary conditions

Equations (1)-(3) are numerically integrated in a two-dimensional domain. A horizontal cylinder of diameter \(D = 6\) mm is centered in a square plane of the same size as the
square cross-section of the experimental enclosure of Fig. 3. No-slip boundary conditions are applied at the walls of the square region (i.e., \( \nu_w = 0 \)) and on the surface of the circular source pipe (i.e., \( \nu_s = 0 \)) as shown in Fig. 1. Isothermal boundary conditions are implemented at the walls of the cavity. This is a more accurate representation of the experimental reality, because it takes into account the finite flux of heat through the walls. Initially, the square region is filled with 508308 uniformly spaced particles at rest, where the fluid is represented by particles that carry the properties of glycerin at 20°C, i.e., \( \rho = 1264 \, \text{kg} \, \text{m}^{-3} \), \( c_p = 2368 \, \text{J} \, \text{kg}^{-1} \, \text{K}^{-1} \), \( k = 0.286 \, \text{W} \, \text{m}^{-1} \, \text{K}^{-1} \), \( \eta = 1.519 \, \text{kg} \, \text{m}^{-1} \, \text{s}^{-1} \), and \( \beta = 5.97 \times 10^{-4} \, \text{K}^{-1} \). These quantities yield the Prandtl number value \( Pr = 12671.05 \). With this spatial resolution the inter-particle separation distance is 0.7 mm in both the \( x \)- and \( y \)-directions, while the smoothing length is set to \( \approx 0.99 \) mm. Incompressibility of the fluid is guaranteed by setting \( c_0 = 50 \, \text{m} \, \text{s}^{-1} \) in Eq. (7).

No-slip boundary conditions are implemented at the walls of the vessel using the method of dynamic boundary particles developed by Crespo et al. [46]. In this method, a linear distribution of uniformly-spaced particles is placed at the walls of the enclosure, with separations of \( h/1.42 \). A second line of uniformly-distributed particles is placed outside the computational domain at distances \( \Delta x/2 \) and \( \Delta y/2 \) from the wall particles so that they are arranged in a staggered grid. This external particles are used to cope with the problem of kernel deficiency outside the computational domain. The wall particles are updated using the same loop as the inner fluid particles and so they are forced to satisfy Eqs. (7) and (9). However, they are not allowed to move according to Eqs. (10) and (12) so that their initial positions and velocities \( \nu_w = 0 \) remain unchanged in time. In this way, the presence of the wall is modelled by means of a repulsive force, which is derived from the source term of the momentum Eq. (10) and includes the effects of compressional, viscous, and gravitational forces. This force is exerted by the wall particles on the approaching fluid particles only when the latter get closer than a distance \( d = 2h \) from the wall. Across the wall, a particle \( a' \) is assigned a mass \( m_{a'} = m_a \), and a density \( \rho_{a'} = \rho_w \), while in order to assure Neumann boundary conditions for the pressure \( \rho_{a'} = \rho_w \) (where the subscript \( w \) indicates wall properties). With a number of 508308 fluid particles, 2917 linearly distributed particles are needed outside the computational domain, giving a total number of 511225 SPH particles. Isothermal wall boundary conditions are specified by setting \( T_{a'} = 2T_w - T_a \) [25], where the wall temperature, \( T_w \), is initially assumed to be in thermal equilibrium with the glycerin at 20°C and \( T_a \) is the temperature of fluid particle \( a \) when it is at a distance \( < 2h \) from the wall.

The circle at the center of the square cavity in Fig. 1, representing the surface of the cylindrical pipe, is also made of fixed boundary particles carrying a temperature \( T_c \). Six model calculations are considered where the value of \( T_c \) is varied above and below the initial fluid temperature \( T_0 = 20°C \) (293.15 K).

3 Experimental methods

In order to validate the results provided by the SPH simulations, detailed measurements of the natural convection flow were obtained experimentally.

3.1 Test apparatus

A slender acrylic enclosure of height \( H = 0.5 \) m, length \( L = 0.5 \) m, and width \( W = 0.025 \) m, filled with glycerin (Pr= 12671.05 at 20°C) as the working fluid, was employed for the experiments. The walls of the enclosure thus formed were 6 mm thick to prevent possible optical distortions (Figs. 2 and 3). A horizontal pipe with a circular cross section of diameter \( D = 0.006 \) m spanned the width of the rectangular cavity. Its symmetry axis passed through the points \((H/2,L/2)\) located on the front and back walls of the cavity, as shown in Fig. 3. The pipe was part of a closed circuit which allowed the circulation of water. The temperature of water was tightly controlled by means of a thermal bath within an accuracy interval of \( \pm 0.001 \)°C. Such an arrangement enabled an approximately uniform and steady temperature difference (\( \Delta T \)) between the horizontal pipe and the glycerin within the cavity. The temperature difference was measured every four seconds with five type \( T \) thermocouples (Tc), where four of them were installed on the back wall with a perpendicular orientation with respect to the pipe, while the other one was attached directly to the pipe’s wall (see Fig. 3). The measurements from the thermocouples were acquired by means of a Digi Sense Scanning Thermometer Cole Parmer (Mod. 92000-00) temperature scanner.

The hydrodynamical characteristics of the flow were determined with a Particle Image Velocimetry (PIV) technique, which enabled an accurate spatial and temporal resolution of the velocity field. All measurements were carried out on a plane, perpendicular to the axis of symmetry of the pipe, generated with a laser light sheet. A 1200 mJ pulsating Nd: YAG Laser, with a pulse duration of 4 ms and a wavelength varying from 1064 nm to 582 nm, was used in this study. Images were recorded with a Hisense MKII camera (with an image resolution of 1260 \times 1024 pixels) equipped with a 60 mm Nikon lens and a 532 nm filter. The glycerin was seeded with polyamide (spherical) tracer particles, with a mean diameter of 20 \( \mu \)m and relative density \( \rho_{glycerin}/\rho_{water} \approx 1 \). The Dantec Dynamics Studio software was employed to process the acquired images.

3.2 Experimental procedure

The cell was filled with a prepared mixture of glycerin and tracer particles. The seeding concentration was optimized to yield a high statistical correlation for the displacement of particles in the domain. To achieve a detection probability of at least 90%, the mean number of tracers per interrogation window was set to be greater than 15 [47]. The laser intensity and exposure were adjusted to generate sufficient light scattering by the tracers and guarantee a good signal quality and flow traceability. The cameras were focused and image calibration was attained using a ruler placed in the center plane illuminated by the laser source. In
the experiments, the temperature difference was defined as \( \Delta T = T_{\text{bath}} - T_{\text{glycerin}} \). As it is common practice in most natural convection experiments, we first considered heating the horizontal pipe with \( \Delta T \) values between 0.1 and 10 K. We then considered the opposite case, where the pipe was cooled with temperature differences in the interval \((-10 \, \text{K}, -0.1 \, \text{K})\). Each experiment was repeated three times to check its reproducibility.

Initially, a two-valve bypass prevented the flow of water inside the pipe. The water was thus forced to circulate through the bath, until it acquired a state of thermal equilibrium at \( T_{\text{bath}} \). At this point, both valves were opened to establish the flow inside the pipe and initiate the heat transfer process with the glycerin. All experiments were performed at a room temperature of 20°C.

3.3 Reproducibility of the experimental measurements

The experimental reproducibility was assessed by repeating all experiments three times. Figure 4 shows steady-state profiles of the velocity projected along the line contained in the \( z = 0 \) plane that passes through the center of the heat source. In particular, the plot shows the velocity as a function of height for the case when \( \Delta T = 10 \, \text{K} \) and \( R_a = 242 \) for three different measurements starting from identical conditions, namely Test 1, Test 2, and Test 3. The vertical velocity component is very sensitive to small changes in the initial conditions and, therefore, it is a convenient parameter to measure the reproducibility of the experiments. The form of the profiles is consistent with a numerical prediction by Lauriat and Desrayaud for laminar buoyancy-induced flows above a horizontal thin cylinder immersed in an air-filled vessel. A non-zero velocity exists close to the cylindrical surface as expected when the pipe behaves as a heat source. In terms of the vertical velocity component the differences between the three experimental results are small throughout the entire profiles. The maximum velocity for Test 1 is \( v_{\text{max}} \approx 5.24 \times 10^{-4} \, \text{m s}^{-1} \) and occurs at a height \( y \approx 0.344 \, \text{m} \) above the source. For comparison, the maximum velocities for Tests 2 and 3 are, respectively, \( v_{\text{max}} \approx 5.22 \times 10^{-4} \, \text{m s}^{-1} \) (at height \( y \approx 0.362 \, \text{m} \)) and \( 5.25 \times 10^{-4} \, \text{m s}^{-1} \) (at height \( y \approx 0.350 \, \text{m} \)).

We may then calculate the differences between the profiles of Test 2 and 3, with respect to Test 1, by utilizing the root-mean-square error (RMSE) as a metric. The approximate errors are \( \approx 2.7 \times 10^{-5} \, \text{m s}^{-1} \) between the profiles 1 and 2 and \( \approx 2.6 \times 10^{-5} \, \text{m s}^{-1} \) between the profiles 1 and 3. These results demonstrate that the experiments are reproducible with good accuracy.

4 Validation of the SPH simulations

4.1 Convergence to the experimental data

From the numerical point of view, a particle-number independence test was necessary to determine the quality of the numerical solution as the number of SPH particles is increased. The experimental profile corresponding to Test 1 was selected as a validation standard. Figure 5 illustrates the quality of the numerical results as the number of particles is increased. The initial inter-particle separations at four different resolutions are 1.55 mm for \( N = 104976 \) (dotted line), 1 mm for \( N = 251001 \) (dot-dashed line), 0.7 mm for \( N = 511225 \) (dashed line), and 0.5 mm for \( N = 10002001 \) (solid line). The numerical profiles show an asymptotic tendency to globally converge as the number of particles is increased. In terms of the RMSE metric the errors between the numerical and the experimental data for Test 1 are approximately \( 5.63 \times 10^{-5} \, \text{m s}^{-1} \) (for \( N = 104976 \)), \( 3.25 \times 10^{-5} \, \text{m s}^{-1} \) (for \( N = 251001 \)), \( 3.01 \times 10^{-5} \, \text{m s}^{-1} \) (for \( N = 511225 \)), and \( 2.83 \times 10^{-5} \, \text{m s}^{-1} \) (for \( N = 10002001 \)). On the other hand, the maximum velocities and heights are \( v_{\text{max}} \approx 5.10 \times 10^{-4} \, \text{m s}^{-1} \), \( y \approx 0.350 \, \text{m} \) (for \( N = 104976 \)), \( v_{\text{max}} \approx 5.15 \times 10^{-4} \, \text{m s}^{-1} \), \( y \approx 0.355 \, \text{m} \) (for \( N = 251001 \)), \( v_{\text{max}} \approx 5.20 \times 10^{-4} \, \text{m s}^{-1} \), \( y \approx 0.365 \, \text{m} \) (for \( N = 511225 \)), and \( v_{\text{max}} \approx 5.21 \times 10^{-4} \, \text{m s}^{-1} \), \( y \approx 0.365 \, \text{m} \) (for \( N = 10002001 \)). The curves for \( N = 511225 \) and \( N = 10002001 \) are seen to overlap with a RMSE deviation of \( 5.05 \times 10^{-6} \, \text{m s}^{-1} \). On the basis of these results, a spatial resolution resulting from \( N = 511225 \) particles was adopted for all subsequent simulations conducted in the present study. At this spatial resolution, the numerical errors between the numerical and experimental profiles are observed to fall within the experimental uncertainty (see Fig. 4). The robustness of the method is further confirmed by the fact that these errors decay faster than the characteristic timescale of the heat diffusion process.

4.2 Lid-driven cavity

The SPH scheme is further validated against the lid-driven cavity test. This test has been used as a benchmark for convection heat transfer problems (see Hopp-Hirschler et al. and references therein). The test consists of a circular, isothermal flow which sets in within a square cavity as a result of a constant velocity on the top wall of the cavity. A schematic of the cavity is shown in Fig. 6. We use the same parameters as in Hopp-Hirschler et al., i.e., the side length of the cavity is \( L = 1 \, \text{mm} \), while the fluid density and dynamic viscosity are \( \rho = 1000 \, \text{kg m}^{-3} \) and \( \eta = 0.01 \, \text{Pa·s} \), respectively. Three different calculations are considered for varying velocity of the top wall of the cavity, i.e., \( v_{w} = 0.1 \, \text{m s}^{-1} \), \( 1 \, \text{m s}^{-1} \), and \( 10 \, \text{m s}^{-1} \), corresponding to Reynolds numbers of \( \text{Re} = \rho L v_{w} / \eta = 10, 100, \) and \( 1000 \), respectively. No-slip boundary conditions are applied at the walls of the cavity, while Neumann boundary conditions are employed for the pressure at the walls. At the bottom corners of the cavity the pressure gradients are either very low or even zero. Therefore, in order to avoid the onset of spurious pressures at the bottom corners, the reference pressure there is set to zero.

Figure 7 shows the dimensionless \( y \)-component of the velocity, \( v_{y} / v_{w} \), as a function of the dimensionless horizontal coordinate position, \( x / L \), at \( y / L = 0.5 \) (left column of plots) and the dimensionless vertical coordinate position, \( y / L \), as a function of the dimensionless \( x \)-component of the velocity, \( v_{x} / L \), at \( x / L = 0.5 \) (right column of plots) for \( \text{Re} = 10, 100, \) and \( 1000 \). The SPH profiles at a spatial resolution of...
\( N = 240 \times 240 \) particles are compared with the OpenFOAM simulations of Hopp-Hirschler et al. [29] using a mesh of 240 \times 240 regular cells. The results from these simulations are taken as reference solutions for comparison with the SPH calculations. The velocity profiles are very well reproduced by the present SPH scheme. Using as an indicator of convergence the position of the center of the vortex, the relative errors of the velocity profiles at this position between the \( N = 240 \times 240 \) SPH calculations and the reference OpenFOAM solutions are \( (\varepsilon_u = 0.57\%, \varepsilon_v = 0.49\%) \) for \( \text{Re} = 10 \), \( (\varepsilon_u = 0.25\%, \varepsilon_v = 0.30\%) \) for \( \text{Re} = 100 \), and \( (\varepsilon_u = 0.59\%, \varepsilon_v = 0.34\%) \) for \( \text{Re} = 1000 \). For comparison, Hopp-Hirschler et al. reported for this test relative errors of \( (\varepsilon_u = 0.60\%, \varepsilon_v = 0.42\%) \) for \( \text{Re} = 10 \), \( (\varepsilon_u = 0.24\%, \varepsilon_v = 0.31\%) \) for \( \text{Re} = 100 \), and \( (\varepsilon_u = 0.60\%, \varepsilon_v = 0.32\%) \) for \( \text{Re} = 1000 \) between their \( N = 240 \times 240 \) SPH results and the OpenFOAM solution.

5 Results

In a rather global sense, the flow within the enclosure is set in motion under the action of the buoyant forces, which are induced by the temperature gradients established between the heat source and the surrounding fluid. When the temperature of the source is higher than that of the glycerin (i.e., \( \Delta T > 0 \)), a vertical ascending flow takes place in the upper part of the cavity. In contrast, when the temperature difference is \( \Delta T < 0 \), a descending flow occurs occupying the bottom part of the cavity. Here we consider values of \( \Delta T \) as high as 10 K. However, for values this large the Boussinesq approximation is still valid because \( \beta \Delta T = 5.97 \times 10^{-3} < 1 \), where \( \beta = 5.97 \times 10^{-4} \text{K}^{-1} \) for glycerin at 20°C.

5.1 Flow field visualization

Flow visualization was carried out by recording the displacement of the tracers within the cell. The first and third rows of Fig. 8 display the experimental velocity maps (first row) and their corresponding streamlines (third row) for \( \Delta T = 10 \text{K (Ra = 242), \Delta T = 5 \text{K (Ra = 60.8), and \Delta T = 1 K (Ra = 15.2). For comparison, the second and fourth rows show the corresponding fields produced by the SPH simulations for the same temperature differences. For all three temperature differences, an ascending stationary flow forms, which consists of a pair of counter-rotating vortices filling the upper part of the cavity just above the heated cylinder. The size and shape of the vortices depend on the temperature difference and Rayleigh number. From the last two rows of Fig. 8 it is evident that the vortices grow in size with the temperature difference. The red regions in the velocity maps correspond to zones of maximum velocity. In these regions the flow is vertically ascending (i.e., it moves away from the heat source) and the flow streamlines (experiments) and velocity vectors (SPH calculations) are closer together. As was previously found by Cesini et al. [5] using air at atmospheric pressure as the working fluid and Atmane et al. [11] using water, the flow pattern on the upper part of the cavity is characteristic of naturally convected heat transfer from a heated circular cylinder. The circulations expand in the upper part of the cavity, and as the temperature difference increases, both the buoyancy driven flow pattern and convection heat transfer increase. While the convective flow is confined in the upper cavity, heat transfer by conduction prevails at the bottom. Apart from minor details, the SPH calculations reproduce other relevant features of the experimentally obtained flow patterns. For instance, with an increase of the Rayleigh number, the density variation becomes greater, causing a more elongated thermal plume (the yellow and red regions in both the experimental and SPH velocity maps).

![Figure 9](https://example.com/fig9.png)

Table 1. RMSE errors between the experimental and the SPH profiles of the vertical velocity component in Figs. 9 and 12

| \( y/H \) | \( \Delta T = 10 \text{K} \) | \( \Delta T = 5 \text{K} \) |
|---|---|---|
| \( \text{RMSE} \) | \( \text{RMSE} \) |
| \( \text{m s}^{-1} \) | \( \text{m s}^{-1} \) |
| \( \text{m s}^{-1} \) | \( \text{m s}^{-1} \) |
| \( 0.52 \) | \( 3.51 \times 10^{-5} \) | \( 2.86 \times 10^{-5} \) |
| \( 0.68 \) | \( 4.58 \times 10^{-5} \) | \( 1.99 \times 10^{-5} \) |
| \( 0.76 \) | \( 3.41 \times 10^{-5} \) | \( 1.95 \times 10^{-5} \) |
| \( 0.84 \) | \( 2.44 \times 10^{-5} \) | \( 1.12 \times 10^{-5} \) |
| \( 0.92 \) | \( 2.35 \times 10^{-6} \) | \( 8.52 \times 10^{-6} \) |

![Figure 10](https://example.com/fig10.png)

For \( \Delta T = -5 \text{K} \), the relative errors of the vertical velocity profiles at this position between the \( N = 240 \times 240 \) SPH calculations and the reference OpenFOAM solutions are \( (\varepsilon_u = 0.34\%, \varepsilon_v = 0.28\%) \) for \( \text{Re} = 10 \), \( (\varepsilon_u = 0.42\%, \varepsilon_v = 0.31\%) \) for \( \text{Re} = 100 \), and \( (\varepsilon_u = 0.50\%, \varepsilon_v = 0.32\%) \) for \( \text{Re} = 1000 \) between their \( N = 240 \times 240 \) SPH results and the OpenFOAM solution.
the agreement between the experimental and SPH velocity profiles of Fig. 9 is measured in terms of the RMSE metric. These values are listed in Table 1. The lowest errors in both cases always occur when \( y/H = 0.52 \), i.e., close to the heat source. At greater heights, the RMSE errors increase marginally. However, in all cases the errors are always less than \( 5 \times 10^{-5} \text{ m s}^{-1} \).

Figure 10 shows the profiles of the vertical component of the velocity along a vertical line in the \( z = 0 \) plane passing through the center of the source for temperature differences of 1, 5, and 10 K. These profiles correspond to the velocity field inside the red plumes shown in Fig. 8. In all cases, the magnitude of the velocity achieves maximum values in the inner region of the plume close to the source \((0.6 < y/H < 0.7)\). As the temperature difference increases, the region of maximum velocity becomes progressively broader. This observation is consistent with the red plumes of Fig. 8 that become more elongated for higher temperature differences. A non-zero velocity exists at the heat source \((y/H = 0.5)\). This feature, as well as the form of the profiles shown in Fig. 10, are in good agreement with the predictions of Lauriat and Desrayaud [3], who reported 2D calculations for the time-dependent laminar buoyancy-induced flow above a horizontal line heat source immersed in an air-filled vessel. The SPH profiles match the experimental data with RMSEs of \( \approx 1.4 \times 10^{-4} \text{ m s}^{-1} \) for \( \Delta T = 10 \) and 5 K, and of \( \approx 8.1 \times 10^{-5} \text{ m s}^{-1} \) for \( \Delta T = 1 \text{ K} \). Further calculations with lower spatial resolution (not reported here) show that the agreement deteriorates particularly around the maximum. Therefore, we expect that by increasing the resolution above \( N = 511225 \) particles the quality of the results will improve. From Figs. 8 and 9 it follows that the steady-state flow patterns are not exactly symmetric with respect to the central vertical axis \((x = 0)\). Asymptotic flow predictions by Angeli et al. [12] indicate that steady non-symmetric temperature and velocity fields may sometimes appear in double-cell circulation patterns for different ratios of the cylinder’s diameter to the cavity’s side and low Ra-values.

When \( \Delta T < 0 \), the cylindrical source is colder than the surrounding glycerin, thereby producing a vertically descending flow. A pair of counter-rotating vortices forms again, which expands in the bottom part of the cavity. Figure 11 shows the details of the flow for \( \Delta T = -5 \text{ K} \) (Ra = 93.8) and \( \Delta T = -1 \text{ K} \) (Ra = 15.7). The counter-rotating vortices are nearly symmetrical and advect glycerin from the bulk of the fluid located beneath the cylinder. A vertically descending flow with a maximum velocity of \( -4.60 \times 10^{-4} \text{ m s}^{-1} \) (experiment) and \( -4.61 \times 10^{-4} \text{ m s}^{-1} \) (SPH) takes place in the red region within the plume. The fine details of the flow are reproduced by the SPH simulations, because the velocity vectors closely follow the experimentally obtained streamlines. When the cooler glycerin reaches the bottom surface of the cavity, it splits up into two nearly symmetric cells occupying the entire lower half of the enclosure. This can be clearly observed in the experimental and SPH velocity maps of Fig. 11.

The profiles of the vertical velocity component at different heights from the bottom surface of the cavity are depicted in Fig. 12 for the descending flows shown in Fig. 11. The SPH solution (solid lines) is compared with the experimental data (symbols). Notice how the descending flow velocity always reaches a maximum just below the source. At either side of the source, the velocity decays sharply and reverts to positive values close to the lateral surfaces of the cavity. In contrast to the cases where \( \Delta T > 0 \), the strength of the flow is more sensitive to the temperature difference. For instance, when \( \Delta T = -1 \text{ K} \), the inverted peaks become much less pronounced and the ascending flow near the lateral walls of the cavity is practically non-existent. Table 1 lists the RMSEs of the SPH solutions along with the experimental data. In all cases, the errors are always lower than \( 6.28 \times 10^{-5} \text{ m s}^{-1} \).

Another comparison between the SPH and the measured maximum values of the vertical velocity components is illustrated in Fig. 13, for all positive and negative temperature differences. Positive values of the velocity correspond to the ascending jet produced by a positive temperature difference, while negative values of the velocity are associated with negative temperature differences. The numerically predicted maximum velocities are in reasonably good agreement with the experimentally obtained values. The actual difference between both curves in terms of the RMSE is \( 8.41 \times 10^{-6} \text{ m s}^{-1} \).

6 Conclusions

Natural convection heat transfer from a horizontal, small-diameter cylinder in a glycerin-filled slender cavity of square-cross section was investigated experimentally and numerically with the aid of a weakly compressible smooth particle hydrodynamics (WCSPH) scheme. Both the experiments and the numerical calculations were carried out for positive (0.1 \( \leq \Delta T \leq 10 \text{ K} \)) and negative (\(-10 \leq \Delta T \leq -0.1 \text{ K} \)) temperature differences between the source and the surrounding fluid, corresponding to low Rayleigh numbers between \( \approx 1.18 \) and \( \approx 242 \).

Use of the PIV technique allowed to determine the velocity fields for the convection flow around the cylindrical source along the vertical centerline normal to the cylinder axis. For \( \Delta T > 0 \), a pair of counter-rotating vortices is formed, which occupies the upper part of the cavity above the source and grows in intensity for increasing temperature differences (or equivalently, increasing values of the Rayleigh number). The opposite is observed when \( \Delta T < 0 \), where now the two circulation zones are confined in the bottom part of the cavity, which also grow in intensity as the magnitude of the temperature difference increases. When \( \Delta T > 0 \), the flow pattern just above the source is characterized by a plume of vertically ascending flow, while the converse occurs when \( \Delta T < 0 \), where the flow is now vertically descending below the source. In both cases, the maximum vertical velocities always occur near the source. The SPH simulations using the same experimental conditions predicted maximum velocities that are in very good agreement with the experimentally observed values. The main features of the flow patterns are qualitatively well reproduced by the numerical calculations. Natural convection under the present geometrical conditions
will be more deeply investigated for larger magnitudes of the temperature difference and varied widths of the cavity.

Acknowledgements

We are grateful to the reviewers who have raised a number of suggestions and comments that have improved the content of the manuscript. F.A. acknowledges the Fondo Sectorial CONACYT - Secretaría de Energía for financial support from the Hidrocarburos Estancias Posdoctorales Program of Mexico. C.E.A.-R. thanks financial support from CONACYT under the Project No. 368. C.E.A.-R. is a research fellow commissioned to the University of Guanajuato (under Project No. 368). We acknowledge financial support from the European Union’s Horizon 2020 Programme under ENERXICO Project, grant agreement No. 828947, and under the Mexican CONACYT-SENER-Hidrocarburos, grant agreement No. B-S-69926. The calculations of this paper were performed using the supercomputing facilities of Abacus-Laboratorio de Matemática Aplicada y Cómputo de Alto Rendimiento of Cinvestav-IPN and the Barcelona Supercomputing Center ( MareNostrum).

References

[1] Zhang, X., Zhang, Y., Liu, Z., and Liu, J., 2020. “Analysis of heat transfer and flow characteristics in typical cambered ducts”. *International Journal of Thermal Sciences*, vol. 150, p. 106226.

[2] Deschamps, V., and Desrayaud, G., 1994. “Modeling a horizontal heat-flux cylinder as a line source”. *Journal of Thermophysics and Heat Transfer*, vol. 8(1), pp. 84–91.

[3] Lauriat, G., and Desrayaud, G., 1994. “Buoyant plane plumes from heated horizontal confined wires and cylinders”. *Sadhana*, vol. 19(5), pp. 671–703.

[4] Liñán, A., and Kurdyumov, V. N., 1998. “Laminar free convection induced by a line heat source, and heat transfer from wires at small grashof numbers”. *Journal of Fluid Mechanics*, vol. 362, pp. 199–227.

[5] Cesini, G., Paroncini, M., Cortella, G., and Manzan, M., 1999. “Natural convection from a horizontal cylinder in a rectangular cavity”. *International Journal of Heat and Mass Transfer*, vol. 42(10), pp. 1801–1811.

[6] Kuehn, T. H., and Goldstein, R. J., 1980. “Numerical solution to the navier-stokes equations for laminar natural convection about a horizontal isothermal circular cylinder”. *International Journal of Heat and Mass Transfer*, vol. 23(7), pp. 971–979.

[7] Farouk, B., and Gücü, S. I., 1981. “Natural convection from a horizontal cylinder – laminar regime”. *Journal of Heat Transfer*, vol. 103(3), pp. 522–527.

[8] Fujii, T., Fujii, M., and Honda, T., 1982. “Theoretical and experimental studies of free convection around a long horizontal thin wire in air”. In *Proceedings of the 7th International Heat Transfer Conference*, Vol. 6, *International Heat Transfer Conferences (IHTC)*, IHTC Digital Library, pp. 311–316.

[9] Karim, F., Farouk, B., and Namer, I., 1986. “Natural convection heat transfer from a horizontal cylinder between vertical confining adiabatic walls”. *Journal of Heat Transfer*, vol. 108(2), pp. 291–298.

[10] Saitoh, T., Sajiki, T., and Maruhara, K., 1993. “Bench mark solutions to natural convection heat transfer problem around a horizontal circular cylinder”. *International Journal of Heat and Mass Transfer*, vol. 36(5), pp. 1251–1259.

[11] Atmane, M. A., Chan, V. S. S., and Murray, D. B., 2003. “Natural convection around a horizontal heated cylinder: The effects of vertical confinement”. *International Journal of Heat and Mass Transfer*, vol. 46(19), pp. 3661–3672.

[12] Angeli, D., Levoni, P., and Barozzi, G. S., 2008. “Numerical predictions for stable buoyant regimes within a square cavity containing a heated horizontal cylinder”. *International Journal of Heat and Mass Transfer*, vol. 51(3–4), pp. 553–565.

[13] Kitamura, K., Kami-iwa, F., and Misumi, T., 1999. “Heat transfer and fluid flow of natural convection around large horizontal cylinders”. *International Journal of Heat and Mass Transfer*, vol. 42(22), pp. 4093–4106.

[14] Misumi, T., Suzuki, K., and Kitamura, K., 2003. “Fluid flow and heat transfer of natural convection around large horizontal cylinders: Experiments with air”. *Heat Transfer – Asian Research*, vol. 32(4), pp. 293–305.

[15] Reymond, O., Murray, D. B., and O’Donovan, T. S., 2008. “Natural convection heat transfer from two horizontal cylinders”. *Experimental Thermal and Fluid Science*, vol. 32(8), pp. 1702–1709.

[16] Koizumi, H., and Hosokawa, I., 1996. “Chaotic behavior and heat transfer performance of the natural convection around a hot horizontal cylinder affected by a flat ceiling”. *International Journal of Heat and Mass Transfer*, vol. 39(5), pp. 1081–1091.

[17] Ghaddar, N. K., 1992. “Natural convection heat transfer between a uniformly heated cylindrical element and its rectangular enclosure”. *International Journal of Heat and Mass Transfer*, vol. 35(10), pp. 2327–2334.

[18] Ghaddar, N. K., and Thiele, F., 1994. “Natural convection over a rotating cylindrical heat source in a rectangular enclosure”. *Numerical Heat Transfer, Part A: Applications*, vol. 26(6), pp. 701–717.

[19] De, A. K., and Dalal, A., 2006. “A numerical study of natural convection around a square, horizontal, heated cylinder placed in an enclosure”. *International Journal of Heat and Mass Transfer*, vol. 49(23-24), pp. 4608–4623.

[20] Kim, B. S., Lee, D. S., Ha, M. Y., and Yoon, H. S., 2008. “A numerical study of natural convection in a square enclosure with a circular cylinder at different vertical locations”. *International Journal of Heat and Mass Transfer*, vol. 51(7-8), pp. 1888–1906.

[21] Adnani, M., Meziani, B., Ourrad, O., and Zitoune, M., 2016. “Natural convection in a square cavity: Numerical study for different values of prandtl number”. *Fluid Dynamics and Materials Processing*, vol. 12(1), pp. 1–14.

[22] Cleary, P. W., 1998. “Modelling confined multi-material heat and mass flows using sph”. *Applied Math-
ematical Modelling, 22(12), pp. 981–993.
[23] Cleary, P. W., and Monaghan, J. J., 1999. “Conduction modelling using smoothed particle hydrodynamics”. Journal of Computational Physics, 148(1), pp. 227–264.
[24] Moballa, B., Chern, M.-J., and Odhiambo, E., 2013. “Incompressible smoothed particle hydrodynamics modeling of thermal convection”. Interaction and Multiscale Mechanics, 6(2), pp. 211–235.
[25] Szwecl, K., Pozorski, J., and Tanière, A., 2011. “Modeling of natural convection with smoothed particle hydrodynamics: Non-boussinesq formulation”. International Journal of Mass and Heat Transfer, 54(23-24), pp. 4807–4816.
[26] Yang, X., and Kong, S.-C., 2019. “Numerical study of natural convection in a horizontal concentric annulus using smoothed particle hydrodynamics”. Engineering Analysis with Boundary Elements, 102, pp. 11–20.
[27] Garoosi, F., and Shakibaenia, A., 2020. “Numerical simulation of entropy generation due to natural convection heat transfer using kernel derivative-free (kdf) incompressible smoothed particle hydrodynamics (isph) model”. International Journal of Mass and Heat Transfer, 150, p. 119377.
[28] Garoosi, F., and Shakibaenia, A., 2020. “An improved high-order isph method for simulation of free-surface flows and convection heat transfer”. Powder Technology, 376, pp. 668–696.
[29] Hopp-Hirschler, M., Shadloo, M. S., and Nieken, U., 2018. “A smoothed particle hydrodynamics approach for thermo-capillary flows”. Computers & Fluids, 176, pp. 1–19.
[30] Nasiri, H., Jamalabadi, M. Y. A., Sadeghi, R., Safaei, M. R., Nguyen, T. K., and Shadloo, M. S., 2019. “A smoothed particle hydrodynamics approach for numerical simulation of nano-fluid flows. application to forced convection heat transfer over a horizontal cylinder”. Journal of Thermal Analysis and Calorimetry, 135, pp. 1733–1741.
[31] Hopp-Hirschler, M., Shadloo, M. S., and Nieken, U., 2019. “Viscous fingering phenomena in the early stage of polymer membrane formation”. Journal of Fluid Mechanics, 864, pp. 97–140.
[32] Becker, M., and Teschner, M., 2007. “Weakly compressible sph for free surface flows”. In Proceedings of the 2007 ACM SIGGRAPH/Eurographics Symposium on Computer Animation, ACM SIGGRAPH/Eurographics, ACM Press, San Diego, pp. 209–217.
[33] Angeli, D., Pagona, A., Corticelli, M. A., Fichera, A., and Barozzi, G. S., 2011. “Bifurcations of natural convection flows from an enclosed cylindrical heat source”. Frontiers in Heat and Mass Transfer, 2(2), p. 023003.
[34] Fiscaletti, D., Angeli, D., Tarozzi, L., and Barozzi, G. S., 2013. “Buoyancy-induced transitional flows around an enclosed horizontal cylinder: An experiment”. International Journal of Heat and Mass Transfer, 58(1–2), pp. 619–631.
[35] Riay, L., Fangzheng, W., and Xinwei, W., 2012. “Heating of heavy oil by circulating hot water in closed double casing in ultra-deep wells”. Petroleum Science, 9(4), pp. 506–510.
[36] Zhao, J., Dong, H., Wang, X., and Fu, X., 2017. “Research on heat transfer characteristic of crude oil during the tubular heating process in the floating roof tank”. Case Studies in Thermal Engineering, 10, pp. 142–153.
[37] Gray, D. D., and Giorgini, D. B., 1976. “The validity of the boussinesq approximation for liquids and gases”. International Journal of Mass and Heat Transfer, 19(5), pp. 545–551.
[38] Batchelor, G. K., 1974. Introduction to Fluid Dynamics. Cambridge University Press, Cambridge, UK.
[39] Monaghan, J. J., 1992. “Smoothed particle hydrodynamics”. Annual Review of Astronomy and Astrophysics, 30, pp. 543–574.
[40] Gómez-Gesteira, M., Rogers, B. D., Crespo, A. J. C., Dalrymple, R. A., Narayanaswamy, M., and Domínguez, J. M., 2012. “Sphysics–development of a free-surface fluid solver–part 1: Theory and formulations”. Computers & Geosciences, 48, pp. 289–299.
[41] Liu, M. B., and Liu, G. R., 2010. “Smoothed particle hydrodynamics (sph): An overview and recent developments”. Archives of Computational Methods in Engineering, 17(1), pp. 25–76.
[42] Dehnen, W., and Aly, H., 2012. “Improving convergence in smoothed particle hydrodynamics simulations without pairing instability”. Monthly Notices of the Royal Astronomical Society, 425(2), pp. 1068–1082.
[43] Zhu, Q., Hernquist, L., and Li, Y., 2015. “Numerical convergence in smoothed particle hydrodynamics”. The Astrophysical Journal, 800(1), p. 13 id. 6.
[44] Sigalotti, L. D. G., Rendón, O., Klapp, J., Vargas, C. A., and Cruz, F., 2019. “A new insight into the consistency of the sph interpolation formula”. Applied Mathematics and Computation, 356, pp. 50–73.
[45] Rosswog, S., 2015. “Sph methods in the modelling of compact objects”. Living Reviews in Computational Astrophysics, 1(1), pp. 1–109.
[46] Crespo, A. J. C., Gómez-Gesteira, M., and Dalrymple, R. A., 2007. “Boundary conditions generated by dynamic particles in sph methods”. Computers, Materials and Continua, 5, pp. 173–184.
[47] Keane, R. D., and Adrian, R. J., 1990. “Optimization of particle image velocimeters. part i: Double pulsed systems”. Measurement Science and Technology, 1(11), pp. 1202–1215.
Fig. 1. Schematic of the computational domain and its boundary conditions. The circle at the center represents the cylindrical heat source.

Fig. 2. Elements of the test apparatus.

Fig. 3. Schematic drawing of the test cell. The legend P1 indicates the location of the thermocouple on the pipe surface, which serves to measure the temperature of the pipe $T_s = T_{\text{bath}}$, while P2 provides a measure of the glycerin temperature close to the pipe. The legends P3-P6 indicate the location of the thermocouples on the right side of the back wall of the vessel, which provide a measurement of the glycerin temperature away from the heat source.

Fig. 4. Experimentally obtained profiles for the vertical component of the velocity along a vertical line on the $z = 0$ plane passing through the center of the heat source for three experiments starting with identical conditions and $\Delta T = 10$ K.

Fig. 5. Particle number independence study and comparison with the experimental data of Test 1 (empty circles) for $N = 104976$ (dotted line), $N = 251001$ (dot-dashed line), $N = 511225$ (dashed line), and $N = 10002001$ (solid line) SPH particles.

Fig. 6. Schematical configuration of the lid-driven cavity test problem.

Fig. 7. Velocity profiles of the vertical velocity, $v_y/v_w$, as a function of $x/L$ at $y/L = 0.5$ (left column) and of the horizontal velocity, $v_x/v_w$, as a function of $y/L$ at $x/L = 0.5$ (right column) as obtained for the lid-driven cavity test for $Re = 10, 100, \text{ and } 1000$. The SPH profiles (empty circles) are compared with the reference solution (solid lines) obtained by Hopp-Hirschler et al. [29] using an OpenFOAM solver.

Fig. 8. Experimental velocity maps and streamlines (first and third row) as compared with the SPH simulations (second and last row) for varying temperature differences: $\Delta T = 10$ K ($Ra = 242$; left column), $\Delta T = 5$ K ($Ra = 60.8$; middle column), and $\Delta T = 1$ K ($Ra = 15.2$; right column).

Fig. 9. Profiles of the vertical velocity component $v$ across the full length of the cavity at different heights from the cylindrical source for: $Ra = 242$ ($\Delta T = 10$ K) (left) and $Ra = 60.8$ ($\Delta T = 5$ K) (right). For reference the heat source is at $x/H = 0.5$. The symbols depict the experimentally obtained profiles, while the solid lines correspond to the SPH profiles.

Fig. 10. Profiles of the vertical velocity component $v$ along a vertical line in the $z = 0$ plane passing through the center of the source for temperature differences of 1, 5, and 10 K between the heat source and the surrounding fluid. The experimentally obtained profiles (symbols) are compared with those obtained from the SPH simulations (solid lines).

Fig. 11. Experimental velocity map and streamlines (first and third row) as compared with the SPH simulations (second and last row) for $\Delta T = -5$ K ($Ra = 93.8$; left column) and $\Delta T = -1$ K ($Ra = 15.7$; right column).

Fig. 12. Profiles of the vertical velocity component $v$ across the full length of the cell at different heights from the bottom surface of the cell below the cylindrical source for $\Delta T = -5$ K ($Ra = 93.8$). The experimentally obtained profiles (symbols) are compared with those obtained from the SPH simulations (solid lines). For reference the cylindrical source is at $x/L = 0.5$. 

Fig. 13. Comparison between the experimentally and numerically obtained maximum values of the vertical velocity component for all temperature differences considered.
FIGURE 4
FIGURE 6
FIGURE 7
FIGURE 9
FIGURE 10

$\nu$ (m/s) vs $y / H$

- $\text{SPH, } \Delta T = 10 \text{ K}$
- $\text{Exp, } \Delta T = 10 \text{ K}$
- $\text{SPH, } \Delta T = 5 \text{ K}$
- $\text{Exp, } \Delta T = 5 \text{ K}$
- $\text{SPH, } \Delta T = 1 \text{ K}$
- $\text{Exp, } \Delta T = 1 \text{ K}$
FIGURE 13