BEM Model for Coupled Natural Convection and Radiation in a Participating Compressible Grey Fluid

Peter Črnjac¹, Tadej Črnjac²

¹²Faculty of Mechanical Engineering, University of Maribor, Smetanovaulica 17, SI-2000 Maribor, Slovenia

Abstract—The objective of this article is to develop a boundary element numerical model to solve coupled problems involving heat energy diffusion, convection and radiation in a square differentially heated cavity filled with participating grey medium. The P₁ approximation is used to solve the radiative transfer equation. The governing Navier-Stokes equations are written in the velocity-vorticity formulation for the kinematics and kinetics of the fluid motion. The approximate numerical solution algorithm is based on boundary element numerical model in its macro-element formulation. The developed algorithm is validated by comparing results of radiative heat transfer with benchmark data. Furthermore, the developed algorithm is tested by simulating natural convection and radiation heat transfer under large temperature differences where compressible flow solution is required.

IndexTerms—Boundary element method, fluid flow, heat transfer, radiation, velocity-vorticity.

I. INTRODUCTION

The combined heat transfer of the radiation and natural convection in the participating (absorbing-emitting-scattering) media is significant in many fields of building and industry, including the boiler, furnace, building thermal comfort and solar reactor. The research on the analysis and numerical solution of heat transfer and fluid flow phenomena where radiative heat exchange has an essential contribution, becomes a key aspect for the employment of computational fluid dynamics simulations as a worthwhile complement to experimental research into industry-related problems. These problems involve the solution of the Navier-Stokes equations and radiative transfer equation (RTE).

Radiative heat transfer plays an important role in the heat transfer in cavities [8], [9], [10], [11], [12], [20]. Many investigations dealing with coupling natural convection and the radiation in cavities have been conducted with a transparent medium [18], [20]. Participating gasses with heteropolar diatomic molecules, such as carbon dioxide (CO₂) and water vapour, which emit and absorb thermal radiation, have an important effect on the heat transfer in cavities. However, many real engineering problems involve truly absorbing-emitting gases. In this case, volumetric radiation can significantly affect the temperature field which, in turn, induces changes in the fluid dynamic.

Regarding the coupling of radiation with the double diffusive natural convection, most of the available investigations use the simple assumption of fictitious grey medium. In these works, the fluid was generally regarded as optically thick and the radiative fluxes were calculated by using the Rosseland approximation. Ibrahim et al. [8], Mezrhab et al. [12], and Mouteekir et al. [14] have investigated the coupling phenomena in a gas mixture. They considered a more realistic situation of an absorption coefficient of fluid proportional to the local temperature and concentration of the absorbing species. These studies are still limited to the grey gas assumption.

The objective of this article is to develop a boundary element numerical model (BEM) to solve coupled problems involving heat energy diffusion, convective and radiative heat transfer in a participating fluid [15]. BEM has been previously applied for the solution of heat conduction and coupled heat conduction-convection problems by many authors. The addition of radiation was considered by [2], [3]. However, unlike the previous studies [2], [3] in this study we propose the use of the Grey Gas Model (GGM) [1] for the participating medium.

The developed algorithm is tested by simulating natural convection and radiation heat transfer under large temperature differences where compressible flow solution is required [5], [17]. The impact of radiation on the overall heat transfer is presented using the approach for optical thick fluids, i.e. the P₁ radiative model [6]. In the model, we express the incident radiation at a given position in the radiation field by the nonlinear nonhomogeneous modified Helmholtz equation. Under the Marshak boundary condition, we solve the equation iteratively as a coupled system with the energy equation.

Next, the governing equations are transformed with the use of the velocity-vorticity variables formulation into kinematics and kinetics [2]-[4]. By applying BEM, we transform partial differential equations (PDE) into integral equations [7], [21]. To test the validity and accuracy of the proposed numerical scheme, we study the problem of the overall heat transfer in a closed square cavity filled with a grey participating medium. The velocity and temperature fields together with the total heat transfer are calculated for the Rayleigh numbers in the laminar regime, and the
solutions compared to the published standard results [9], [10], [14].

II. GOVERNING EQUATIONS

The present development is focused on the laminar flow of compressible isotropic radiation semi-transparent fluid in solution domain \( \mathcal{R} = \Omega \times \tau \) where \( \Omega \) stands for the two-dimensional plane domain bounded by boundary \( \partial \Omega \) defined by the outward-pointing unit normal \( n \), whilst \( \tau \) represents time dimension of the transport phenomenon. The mass, momentum and energy equations are given by the following set of nonlinear equations

\[
\begin{align*}
\frac{\partial v_j}{\partial x_j} & = -\frac{1}{\rho} \frac{D\rho}{Dt} = \mathcal{D}, \\
\rho \frac{Dv_i}{Dt} & = -\frac{\partial \tau_{ij}}{\partial x_j} - \frac{\partial p}{\partial x_i} + \rho g_i, \\
c \frac{DT}{Dt} & = -\frac{\partial q^p_j}{\partial x_j} - \frac{\partial q^R_j}{\partial x_j},
\end{align*}
\]

in the Cartesian frame \( x \), where the field functions of interest are velocity vector field \( v_i(\eta, t) \), pressure scalar field \( p(\eta, t) \) and the temperature scalar field \( T(\eta, t) \). \( \rho \) and \( c \) denote variable mass density and isobaric specific heat capacity per unit volume, \( c = \rho c_p \) is time, \( g_i \) is gravitational acceleration vector, \( \mathcal{D} \) represents the divergence of the velocity field or the local expansion rate, whilst the vector variables \( q^p \) and \( q^R \) are heat diffusion and radiation fluxes, respectively. The differential operator \( D(\cdot)/Dt \) stands for the Stokes material derivative.

The conservation equations (2) and (3) contain two molecular diffusive fluxes, i.e. \( \tau_{ij} \) and \( q^p_j \), representing the diffusion of linear momentum and the heat flux vector, respectively. The Newton linear momentum diffusion constitutive model for compressible viscous shear fluid is considered, such as [3]

\[
\tau_{ij} = 2\eta \varepsilon_{ij} - \frac{2}{3} \eta \mathcal{D} \delta_{ij}, \tag{4}
\]

where \( \eta \) is the dynamic viscosity coefficient. For most heat-transfer problems of practical importance, the simplification known as the Fourier law of heat diffusion is accurate enough, namely

\[
q^p_i = -k \frac{\partial T}{\partial x_i}, \tag{5}
\]

where \( k \) is the diffusion thermal heat conductivity.

The governing equation for radiative heat transfer is the radiative transfer equation (RTE) and was discussed previously by Crnjac et al. [2]-[4]. The RTE is based on an energy balance for radiation passing through a differential volume in a participating medium in local thermodynamic equilibrium (LTE). For the coupling of the radiative heat transport with the fluid dynamics, LTE is assumed and the time dependence of the RTE is neglected [13], [19]. It follows from LTE that the temperature of the fluid and the corresponding radiative temperature in the medium are equal. The solution of the RTE implies a considerable computational cost due to the directional nature of the intensity radiation field. This high computational cost limits detail in the simulation of coupled radiation and convection. Therefore, improvements of the numerical methods and the fundamental analysis of these complex phenomena have motivated interest in the scientific community.

The spectral extinction coefficient of the participating medium \( K_\lambda(\mathcal{R}) \) is defined as the sum of the spectral absorption coefficient \( a_\lambda(\mathcal{R}) \) and the spectral scattering coefficient \( s_\lambda(\mathcal{R}) \) [19]. In the grey medium with constant \( a \) and \( K \) the \( B_\lambda \) approximation reduces the RTE into a nonlinear nonhomogeneous modified Helmholtz equation [2]-[4]

\[
\frac{\partial^2 G}{\partial x_j \partial x_j} - \beta G + b = 0, \tag{6}
\]
with $\beta = 3aK$ and temperature dependent nonhomogeneous term $b = 4\beta \sigma T^4$. The spectral incident radiation function $G_\lambda (\vec{r})$ describes the total intensity $i_\lambda$ impinging on a point $\vec{r}$ in the medium from all directions [17]

$$G_\lambda (\vec{r}) = \int_0^{4\pi} i_\lambda (\vec{r}, \vec{z}) d\omega .$$

(7)

Note that the spectral incident radiation divided by the speed of light $G_\lambda /c$ is the spectral radiative energy density at location $\vec{r}$ in the radiation field. The divergence of the radiation flux vector in equation (3) can be expressed as the local radiation source term $S^R$

$$\frac{\partial q^R}{\partial x_j} = -a[G - 4\sigma T^4] = -S^R .$$

(8)

If it is assumed that the walls are diffuse grey surfaces, the equation (6) is solved using Marshak boundary condition [2]-[4]

$$-\frac{1}{3K} \frac{\partial G}{\partial n} \bigg|_w = \frac{\varepsilon_w}{2(1 - \varepsilon_w)} (4\sigma T_w^4 - G_w),$$

(9)

where $\varepsilon_w$ is the emissivity of the wall and the subscript $w$ denotes the value of the indicated variable at the wall.

In this study, the grey gas absorption coefficients for participating gases are correlated by Barlow et al. [1], so the following expression is used to calculate $a$ in units of ($m^{-1}$, $m^{-1}$)

$$a = c_0 + c_1 \left( \frac{1000}{T} \right) + c_2 \left( \frac{1000}{T} \right)^2 + c_3 \left( \frac{1000}{T} \right)^3 +$$

$$+ c_4 \left( \frac{1000}{T} \right)^4 + c_5 \left( \frac{1000}{T} \right)^5 .$$

(10)

Coefficients $c_0, c_1, c_2, c_3, c_4$ and $c_5$ are calculated for different gases as suggested by [1]. These curve fits were generated for temperatures between 300 K and 2500 K and may be very inaccurate outside this range.

III. VELOCITY-VORTICITY FORMULATION OF NAVIER-STOKES EQUATIONS

Substituting equations (4) and (5) for the non-convective heat and momentum fluxes in conservation equations (2) and (3) the following system of nonlinear Navier-Stokes equations for the primitive variables is developed

$$\frac{\partial \eta}{\partial x_j} = -\frac{1}{\rho} \frac{D\rho}{Dt} = \mathcal{D},$$

(11)

$$\rho \frac{Dv_i}{Dt} = -\varepsilon_{ijk} \frac{\partial \eta_{\omega_k}}{\partial x_j} + 2\varepsilon_{ijk} \frac{\partial \eta}{\partial x_j} \omega_k + 2 \frac{\partial \eta}{\partial x_j} \frac{\partial v_i}{\partial x_j} +$$

$$+ \frac{4}{3} \frac{\partial \eta}{\partial x_i} - 2D \frac{\partial \eta}{\partial x_i} - \frac{\partial p}{\partial x_i} + \rho g_i ,$$

(12)

$$c \frac{DT}{Dt} = \frac{\partial}{\partial x_j} \left( k \frac{\partial T}{\partial x_j} \right) + S^R ,$$

(13)
where \( \boldsymbol{\epsilon} \) is the unit tensor. Representing the transport properties of the fluid as a sum of a constant and variable part, i.e. \( \eta = \eta_0 + \tilde{\eta} \), \( k = k_0 + \tilde{k} \), \( c = c_0 + \tilde{c} \) and \( \rho = \rho_0 + \tilde{\rho} \), the momentum and energy equations (12) and (13) can be stated in analogy to the transport equations formulated for the constant transport properties

\[
\frac{D v_i}{Dt} = -\varepsilon_{ijk} v_0 \frac{\partial \omega_k}{\partial x_j} - \frac{1}{\rho_0} \frac{\partial p}{\partial x_i} + \frac{\rho}{\rho_0} g_i + \frac{1}{\rho_0} f_i^m, \quad (14)
\]

\[
\frac{DT}{Dt} = a_0 \frac{\partial^2 T}{\partial x_j \partial x_j} + \frac{S^R}{c_0} + \frac{S^m}{\epsilon_0}, \quad (15)
\]

where the pseudo body force term \( f_i^m \) and pseudo heat source term \( S_T^m \) are introduced into the momentum equation (12) and into energy equation (13) respectively, capturing the variable transport property effects, and given by expressions

\[
f_i^m = -\varepsilon_{ijk} \frac{\partial \tilde{\eta}}{\partial x_j} \omega_k + 2\varepsilon_{ijk} \frac{\partial \tilde{c}}{\partial x_j} \omega_k + 2 \frac{\partial \tilde{c}}{\partial x_j} \frac{\partial v_i}{\partial x_j} +
\]

\[
+ \frac{4 \partial \tilde{\eta} D}{3 \partial x_i} - 2D \frac{\partial \tilde{c}}{\partial x_i} - \tilde{\rho} a_i, \quad (16)
\]

while the pseudo heat source term is given by an expression

\[
S_T^m = \frac{\partial}{\partial x_j} \left( \frac{\tilde{\eta} \frac{\partial T}{\partial x_j}}{\partial x_i} \right) - \frac{\partial T}{\partial x_i} \quad (17)
\]

in which the kinematic viscosity is \( \nu_0 = \eta_0/\rho_0 \), the heat diffusivity \( a_0 = k_0/c_0 \) and the inertia acceleration vector is \( \tilde{a} = D \tilde{v}/Dt \).

In velocity-vorticity formulation the fluid motion computation procedure may be partitioned into its kinetics and kinematics. The kinematics deals with the relationship and restriction among the velocity field at any given instant of time, the vorticity \( \omega_i = \varepsilon_{ijk} \partial v_k / \partial x_j \) and local expansion field at the same instant, and given by the following vector elliptic Poisson equation for the velocity vector

\[
\frac{\partial^2 v_i}{\partial x_j \partial x_j} + \varepsilon_{ijk} \frac{\partial \omega_k}{\partial x_j} \frac{\partial D}{\partial x_i} = 0. \quad (18)
\]

For the known vorticity and local expansion field functions, the corresponding velocity vector can be determined by solving equation (18), providing that appropriate boundary conditions for the velocity are prescribed, i.e. normal and tangential component of the velocity vector.

The kinetic aspect of the fluid motion is governed by the vorticity transport equation

\[
\frac{\partial \omega_i}{\partial t} + \frac{\partial v_j \omega_i}{\partial x_j} = \nu_0 \frac{\partial^2 \omega_i}{\partial x_j \partial x_j} + \frac{\partial \omega_k v_i}{\partial x_j} +
\]

\[
+ \frac{1}{\rho_0} \varepsilon_{ijk} \frac{\partial \rho g_k}{\partial x_j} + \frac{1}{\rho_0} \varepsilon_{ijk} \frac{\partial \tilde{f}_j^m}{\partial x_j}, \quad (19)
\]

describing the redistribution of the vorticity in the fluid domain by different transport phenomena, e.g. diffusion, convection, twisting and stretching, whilst the buoyancy, compressibility, and the nonlinear terms act as a source or strengthen terms [3].

For the two-dimensional plane motion the equations (18) and (19) significantly reduce to plane kinematics given by the following equation
whilst the kinetics is expressed by the scalar vorticity equation

\[ \frac{\partial \omega}{\partial t} + \nabla \times (\mathbf{v} \times \mathbf{D}) = -\nabla p + \mathbf{f}. \]  

To derive the pressure equation, the divergence of equation (14) should be calculated, resulting in the elliptic Poisson pressure equation [3]

\[ \frac{\partial^2 p}{\partial x_i \partial x_i} - \frac{\partial f_{\mathbf{pi}}}{\partial x_i} = 0, \]  

where in the vector function \( f_{\mathbf{pi}} \) the diffusion, inertia, gravitational and nonlinear material property effect are incorporated

\[ \frac{\partial p}{\partial x_i} = f_{\mathbf{pi}} = -\eta_0 \varepsilon_{ij} \frac{\partial \omega_{\mathbf{j}}}{\partial x_i} - \frac{\rho}{\rho_0} \mathbf{a}_i + \rho g_i + f_{\mathbf{m}}. \]  

IV. BOUNDARY-DOMAIN INTEGRAL EQUATIONS

In general, the set of equations (6), (20), (21) and (22) have to be transformed, using the Green identities or weighted residual techniques in combination with appropriate fundamental solutions, into boundary-domain integral equations [2]-[4]. The singular boundary-domain integral representation for the velocity vector can be formulated rendering the following integral formulation for the two-dimensional plane kinematics

\[
\begin{align*}
  c(\xi) \mathbf{v}_i(\xi) + \int_G \mathbf{v}_j q^{*} d\Gamma &= e_{ij} \int_G \mathbf{v}_j q^{*} d\Gamma - e_{ij} \int_\Omega \omega q^{*}_j d\Omega = \\
  -e_{ij} \int_\Omega \omega q^{*}_j d\Omega + \int_\Omega \mathbf{D} q^{*}_j d\Omega,
\end{align*}
\]

where \( q^{*}(\xi, s) \) stands for the elliptic Laplace fundamental solution

\[ u^{*}(\xi, s) = \frac{1}{2\pi} \ln \left( \frac{r}{r_0} \right), \]  

In equation (24) we note \( q^{*} = \partial u^{*}/\partial n \) and \( q^{*}_t = \partial u^{*}/\partial t \) for the normal and tangential derivative of the fundamental solution, while the quantity \( r(\xi, s) \) is the distance vector, which points from the source point \( \xi \) to the field point \( s \). The geometrical coefficient \( c(\xi) \) denotes the fundamental solution related coefficient depending on the position of the source point \( \xi \).

Considering the kinetics in an integral representation one has to take into account parabolic diffusion character of the vorticity transport equation (21). Applying the linear diffusion differential operator the following boundary-domain integral representation corresponding to equation (21) can be derived as

\[
\begin{align*}
  c(\xi) \omega(\xi, \tau_F) + \nu_0 \int_G \int_{\tau_F} \omega q^{*} dt d\Gamma &= \nu_0 \int_G \int_{\tau_{F-1}} \frac{\partial \omega}{\partial n} u^{*} dt d\Gamma \\
  &= \nu_0 \int_G \int_{\tau_{F-1}} \frac{\partial \omega}{\partial n} u^{*} dt d\Gamma.
\end{align*}
\]
where $u^*$ is now parabolic diffusion two-dimensional plane fundamental solution [3]

$$u^*(\xi, t_f; s; t) = \frac{1}{4\pi v_0 \tau} \exp \left[ -\frac{r^2}{4v_0 \tau \tau} \right], \quad (27)$$

where $(\xi, t_f)$ and $(s; t)$ are used for the source and reference space and time field points and $\tau = t_f - t$. Quantities $v_0$, $g_t$ and $f_t^{\text{ext}}$ are the normal velocity component, the tangential component of the gravity and nonlinear source vectors, respectively.

The corresponding integral representation of the pressure equation (22) is given by

$$c(\xi) p(\xi) = \int \int p \dot{q}^* d\Gamma = \int_{t_f}^{t_f} f_p \cdot u_t^* d\Omega, \quad (28)$$

where the vector $f_p^*$ is given by equation (23). For given Neumann boundary conditions, the pressure field is determined with the solution of integral equation (28) taking into account known velocity and vorticity vector field functions, and transport property values.

The integral representation of the nonlinear heat energy diffusion convection transport equation is derived considering the linear parabolic diffusion differential operator yielding the following integral representation [3]

$$c(\xi) \nabla(\xi, t_f) + a_0 \int \int \nabla q^* d\tau =$$

$$= \frac{1}{c_0} \int \int \left( \frac{\partial \nabla}{\partial \tau} - c v_n \nabla \right) u^* d\tau d\Gamma -$$

$$- \frac{1}{c_0} \int \int \left( \nabla \frac{\partial \nabla}{\partial x_j} - c v_j \nabla \right) q_j^* d\tau d\Omega +$$

$$+ \frac{1}{c_0} \int \int \left( T v_j \frac{\partial c}{\partial x_j} + c \nabla - c \frac{\partial \nabla}{\partial t} + S \right) u^* d\tau d\Gamma +$$

$$\int \omega_{t-1} u_{t-1}^* d\Omega,$$
The incident radiation equation (6) is an elliptic modified Helmholtz equation and the corresponding boundary-domain integral representation can be stated as

\[ c(\xi) G(\xi) + \int_{\Gamma} G \frac{\partial \psi}{\partial n} d\Gamma = \int_{\Gamma} 4\beta \sigma T^4 u^* d\Gamma + \int_{\Omega} 4\beta \sigma T^4 u^* d\Omega, \]

where \( u^* \) is now the modified Helmholtz fundamental solution and given by

\[ u^* = \frac{1}{2} K_0(\sqrt{\beta} r) \quad \text{and} \quad q^* = \frac{d_n}{2\pi r^2} \sqrt{\beta} r K_1(\sqrt{\beta} r), \]

whilst \( K_\alpha \) is a modified Bessel function of the second kind of order \( \alpha \).

V. DISCRETIZED INTEGRAL EQUATIONS

For the numerical approximate solution of the field functions, the corresponding integral equations are further written in a discretized form in which the integrals over the boundary and domain are approximated by a sum of the integrals over all boundary elements and over all internal cells. Since the implicit set of equations is written simultaneously for all boundary and internal nodes, this procedure results in a fully populated influence and system matrices, resulting in large computing times and memory demands, which is especially true considering the fluid flow characterised by a high Reynolds number value [3].

In order to improve computational efficiency of the computation we employ the macro-element approach [16]. The idea is to use a collocation scheme for integral equations for each domain cell separately and require that the field functions and their normal derivatives must obey the compatibility and equilibrium conditions over the domain cell boundaries. The final system of equations for the entire domain is then obtained by adding the sets of equations for each macro-element, resulting in a sparse system matrix suitable to solve with iterative techniques.

In our case, each macro-element consists of four continuous 3-node quadratic boundary elements with 1-node constant approximation for the normal flux, and a 9-node continuous quadratic internal cell [3]. The quadratic interpolation functions have been used since a good approximation of the boundary values of the velocity gradients ensure an accurate evaluation of the boundary vorticity values, which strongly influence the stability of the numerical method.

Linear approximation of field functions over each individual time increment is also considered.

VI. TEST EXAMPLES

Details of the geometry and boundary conditions for the simulation of combined natural convection and radiation heat transfer are shown in Fig. 1. In order to check the accuracy of the relevant numerical scheme, a test problem was considered previously analyzed by Lari et al. [9]. It can be observed that the results obtained by Crnjac et al. [3] are in excellent agreement with the previous studies by Lari et al. [9].

![Diagram](image)

Fig.1. The geometry of the cavity: boundary and initial conditions (left), computational mesh (60x60 cells) (right).
The two horizontal walls are perfectly insulated, while the two vertical walls are maintained at two different temperatures $T_h$ and $T_c$, ($T_h > T_c$), respectively. The inner surfaces, in contact with the fluid, are assumed to be grey and diffuse. In all test examples we consider a square cavity ($L = 0.5$ m), filled with a Newtonian compressible viscous semi-transparent fluid. It is submitted to a temperature difference $\Delta T = T_h - T_c$ at the vertical walls, with uniform temperatures $T_h = 600$ K, $T_k = 900$ K and $T_h = 1200$ K, respectively, and $T_c = 300$ K. The top and bottom walls of the enclosure are considered adiabatic, i.e. there is no heat flux through them. The initial conditions are given by the value $T_0 = 450$ K and $p_0 = 101.325$ Pa. On all walls the no-slip condition is imposed for the velocity. The problem is free of any singularity in the boundary conditions except the presence of the corners in the cavity.

The computational mesh is composed of 240 boundary elements and 3600 internal cells, i.e. 60x60 macro-elements with a ratio of 6 between the longest and the shortest element. The convergence criterion was selected as $10^{-7}$. The present time-dependent analyses were performed by running the simulation from the initial state with a time step value of $\Delta T = 1s$.

Because the density of the fluid depends on the temperature, the fluid starts moving upward at the hot wall due to buoyancy, and downward at the cold wall. The velocity of the fluid depends on the Rayleigh number

$$Ra = Pr \frac{g\rho c^2(T_h - T_c)L^2}{\eta^2},$$

where $Pr$ is the characteristic nondimensional Prandtl number, $L$ is the dimension of the square cavity, $T_r$ is a reference temperature defined as $T_r = (T_h + T_c)/2$ and $\rho_r (T_r, p_r)$ is a reference mass density. As a test example, the impact of radiation on the overall heat transfer is analysed by applying the $Pr_{\text{rad}}$ model. In engineering radiative transfer problems, the $Pr_{\text{rad}}$ model should typically be used for spectral optical thicknesses $k_\lambda(\tau) = \int_0^\infty K_\lambda(\tau) d\tau > 1$ [19].

To consider the effect of differing driving forces for buoyant flow, we vary the temperature of the hot wall as the primary variable in this study. As a first numerical example, the impact of radiation on the overall heat transfer is analysed with temperature difference $\Delta T = 300$ K ($Ra = 6.0 \cdot 10^6$). The second computations were performed for $Ra = 3.5 \cdot 10^6$, as a result of imposed $T_h = 900$ K and $T_c = 300$ K.

Fig. 2. Temperature fields for different temperature driving potentials, top row: no radiation, bottom row: $Pr_{\text{rad}}$ model.
example is analysed with temperature difference $\Delta T = 900$ K ($Ra = 2.0 \cdot 10^6$). It was assumed, that the transient simulation results achieved steady state when the selected convergence criteria between two-time steps was satisfied. Let us present the temperature and velocity fields, respectively, predicted by the $P_1$ model for the case of conduction, radiation and significant levels of convection. Fig. 2 compares the influence of the internal radiation with $\kappa_\infty = 10$ on the fluid structure for three temperature differences. It is obvious that the radiation transfer is characteristically different from the convection heat transfer and depends on the properties of the medium including the optical thickness. The effects are visible along the horizontal walls and at the core of the cavity. Compared to the case without radiation, fluid circulation is increased under radiation, and isotherms structures in the cavity are affected by thermal radiation.

![Temperature Profiles](image1)

$
\Delta T = 300$ K  \hspace{1cm}  $\Delta T = 600$ K  \hspace{1cm}  $\Delta T = 900$ K

Fig. 3. Horizontal temperature profiles at $y = L/2$ (top row) and vertical temperature profiles at $x = L/2$ (bottom row), solid line: $P_1$ model, dashed line: without radiation.

![Velocity Fields](image2)
The differences between both cases are clearer in Fig. 3, where the temperature distributions along the x-axis at \( y = L/2 \) and along the y-axis at \( x = L/2 \) are plotted and compared for the same temperature driving potentials. The temperature field predicted with the \( \text{P}_1 \) model is very similar to the benchmark problem studied by Moufekkir et al. [14]. The temperature field of the \( \text{P}_1 \) model through the middle portion of the domain is vertically stratified, with nearly horizontal isotherms, and thermal boundary layers have formed along the walls. It appears from the isotherms that the radiation heat transfer produces a good homogenization of temperature.

Fig. 4. Horizontal \( v_x \) velocity component contours \((\kappa_z = 10)\) for different temperature driving potentials; top row: no radiation, bottom row: \( \text{P}_1 \) model

Fig. 5. Vertical \( v_y \) velocity component contours \((\kappa_z = 10)\) for different temperature driving potentials; top row: no radiation, bottom row: \( \text{P}_1 \) model
Fig. 6. Horizontal velocity component profiles $v_x(x,L/2)$ along $x$-axis at $y = L/2$ (top row) and $v_y(L/2,y)$ along $y$-axis at $x = L/2$ (bottom row) for different temperature driving potentials.

Figs. 4 and 5 show velocity fields for the same temperature driving potentials. The effects of the temperature difference $\Delta T$ on horizontal velocity component profiles $v_x\left(x, \frac{L}{2}\right)$ along $x$-axis at $y = \frac{L}{2}$ and $v_y\left(\frac{L}{2}, y\right)$ along $y$-axis at $x = L/2$ are shown in Figure 6. Vertical velocity component profiles $v_y(x,L/2)$ along $x$-axis at $y = L/2$ and $v_y(L/2,y)$ along $y$-axis at $x = L/2$ for different temperature driving potentials are shown in Figure 7.
Fig. 7. Vertical velocity component profiles $v_x(x, L/2)$ along $x$-axis at $y = L/2$ (top row) and $v_y(L/2, y)$ along $y$-axis at $x = L/2$ (bottom row) for different temperature driving potentials.

It is seen that the maximum of both distributions gradually increases with the temperature difference. Indeed, due to increasing $\Delta T$ the heat transferring in the cavity increases and hence the driven buoyancy force increases. Thus, higher velocities are acquired. It is also seen in figures that the difference between the velocity profile for the $P_1$ model and for the pure convection model at each $\Delta T$ is considerable.

It was found previously that when $Ra$ increases, the natural convection has the great effects on the temperature distribution and heat transfer in the cavity. The diffusion, radiative and overall Nusselt number at the walls are calculated from the heat fluxes as

$$Nu(y) = \frac{L(q^D + q^R)}{k_r(T_h - T_c)} = Nu_D(y) + Nu_R(y), \quad (33)$$

$$Nu_D(y) = \frac{L}{k_r(T_h - T_c)} k^D(T) \left( -\frac{\partial T}{\partial x_w} \right), \quad (34)$$

$$Nu_R(y) = \frac{Lq_w^R}{k_r(T_h - T_c)}, \quad (35)$$

where $k^D(T)$ is temperature dependent heat conductivity and $k_r = k(T_r)$. The temperature dependence of the dynamic viscosity is given by the Sutherland model [2] and the heat conductivity is expressed as $k^D(T) = \eta(T) c_p/Pr$.

Table 1 shows computed diffusion and radiation Nusselt number values, $Nu_D$ and $Nu_R$, for the hot and cold wall as a function of temperature difference for the mesh $M = 60 \times 60$. We can observe that the Nusselt number $Nu_R$ is in the range of $\approx 135-580$ for the hot and $\approx 82-393$ for the cold wall, while the Nusselt number $Nu_D$ is in the range of $\approx 10-14$ for the hot and $\approx 13-24$ for the cold wall. Therefore, $Nu_R = Nu - Nu_D$ is much higher than $Nu_D$ for all cases, which indicates that the radiation heat transfer is dominant heat transfer mechanism in the closed cavity.

<table>
<thead>
<tr>
<th>$\Delta T$</th>
<th>$P_1$ model</th>
<th>cold wall</th>
<th>without radiation</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$Nu_D$</td>
<td>$Nu_R$</td>
<td>$Nu$</td>
</tr>
<tr>
<td>300</td>
<td>10.384</td>
<td>135.7</td>
<td>146.153</td>
</tr>
<tr>
<td>600</td>
<td>10.192</td>
<td>323.0</td>
<td>333.224</td>
</tr>
<tr>
<td>900</td>
<td>10.054</td>
<td>579.3</td>
<td>589.451</td>
</tr>
</tbody>
</table>

VII. CONCLUSION

In this paper we have implemented $P_1$ approximation to simulate combined problems involving conductive, convective and radiative heat transfer in a 2D square cavity filled with a viscous compressible grey fluid. For the $P_1$ model the RTE is a diffusion equation, which is easy to solve with little CPU (Central Processing Unit) demand. The $P_1$ model should typically be used for cases with optical thicknesses larger than one. It should be aware of the following limitations when using the $P_1$ model: $P_1$ model assumes that all surfaces are diffuse. The implementation assumes grey radiation. There may be a loss of accuracy, depending on the complexity of the geometry, if the optical thickness is small.

The main advantage of boundary-element based simulation algorithms is in the fact that a part of the solution of the underlying problem (the fundamental solution) is used to set up the integral formulation of the problem. This enables higher accuracy of the solution of the problem compared to standard techniques such as finite elements or control methods.
volume. This is especially true for diffusion or advection-diffusion type problems, which are considered and which feature high gradients in the solution.

In this paper, we consider optical thickness 10, and discover that at this value the model yielded surprisingly accurate values compared to the published benchmark results. The governing equations of combined heat transfer were solved simultaneously to obtain the temperature and velocity profiles inside the participating medium. However, the model matches very well with the method of discrete ordinates used by Lari et al. [9] to model the radiative heat transfer in optical thick media. Based on the results of this analysis, we recommend the use of the formulation for the buoyant flows in optically thick fluids.

VIII. REFERENCES