

Simulation of steel beam under ceiling jet based on a wind–fire–structure coupling model

Jinggang ZHOU^a, Xuanyi ZHOU^{a*}, Beihua CONG^b, Wei WANG^a, Ming GU^a

^a State Key Laboratory of Disaster Reduction in Civil Engineering, Tongji University, Shanghai 200092, China

^b Shanghai Institute of Disaster Prevention and Relief, Tongji University, Shanghai 200092, China

*Corresponding author. E-mail: zhouxytj@tongji.edu.cn

© Higher Education Press 2023

ABSTRACT For localized fires, it is necessary to consider the thermal and mechanical responses of building elements subject to uneven heating under the influence of wind. In this paper, the thermomechanical phenomena experienced by a ceiling jet and I-beam in a structural fire were simulated. Instead of applying the concept of adiabatic surface temperature (AST) to achieve fluid–structure coupling, this paper proposes a new computational fluid dynamics–finite element method numerical simulation that combines wind, fire, thermal, and structural analyses. First, to analyze the velocity and temperature distributions, the results of the numerical model and experiment were compared in windless conditions, showing good agreement. Vortices were found in the local area formed by the upper and lower flanges of the I-beam and the web, generating a local high-temperature zone and enhancing the heat transfer of convection. In an incoming-flow scenario, the flame was blown askew significantly; the wall temperature was bimodally distributed in the axial direction. The first temperature peak was mainly caused by radiative heat transfer, while the second resulted from convective heat transfer. In terms of mechanical response, the yield strength degradation in the highest-temperature region in windless conditions was found to be significant, thus explaining the stress distribution of steel beams in the fire field. The mechanical response of the overall elements considering the incoming flows was essentially elastic.

KEYWORDS CFD–FEM coupling, steel beam, wind, ceiling jet, numerical heat transfer

1 Introduction

To date, many researchers have analyzed the thermal and mechanical properties of steel components under fire conditions. Compartment fires can generally be divided into the following stages: ignition, early growth, pre-flashover period, flashover, fully developed or post-flashover period, and decay period. In the case of flashover, compartment fires undergo pre-flashover and post-flashover stages. Before a flashover occurs, the fire will develop within the local area of the room, and thus pre-flashover fire can also be called “localized fire”. By contrast, post-flashover fire burns in the whole area of the room, causing greater potential for human harm and building damage. Early studies [1–4] mainly focused on post-flashover fires. In a post-flashover compartment fire, the gas properties within the compartment are

approximately uniform, and thus the compartment ambient temperature can be approximately determined by the time–temperature curve (e.g., ISO-834 curve [5], EC3 European specification [6]). The steel temperature can then be obtained by substituting the fire curve into a one-dimensional (1D) condensed heat-transfer model. However, in actual situations, the threshold of flashover occurrence cannot be reached in large spaces, such as airport terminals, stadiums and parks. In this case, only localized fires will develop [7,8], and thus the uneven heating of building components should be considered.

Some scholars have conducted studies on localized fires, including fire tests. However, realistic localized-fire test reports are quite limited, especially in terms of the response of the structure to non-uniform heating conditions [9]. For example, Kamikawa et al. [10] conducted tests to explore the surface temperature and heat flux of a square steel column adjacent to and surrounded by fire sources. Zhang et al. [11] reported the

results of a thermal test on the heating of a 6 m long steel W beam subjected to a localized fire. Lineham et al. [12] conducted a series of novel fire tests on cross-laminated timber beams subjected to sustained flexural loading. Wiesner et al. [13] described a series of experiments using real waste-bin fires or controlled gas burners placed next to I-section steel columns. Yokobayashi et al. [14] used a propane porous burner to test the steel beam under the ceiling and measured the heat flow and temperature distributions in detail. In the Valencia bridge fire tests, Alos-Moya et al. [15] carried out a series of open-air fire tests under an experimental bridge. Maraveas and Vrakas [16] described in detail the effect of fire on the behavior of concrete. With examples from real fire accidents, they focused on the explosive spalling of concrete. Moreover, Maraveas [17] analyzed the problem of local buckling of steel structures through abundant experimental data and numerical simulations.

The numerical simulation of localized fires involves the coupling of three analytical models, namely, fire analysis, thermal analysis, and structural analysis. A sophisticated computational fluid dynamics (CFD) method is typically used to simulate gas-phase combustion in fire analysis. Heat penetration into the structure is simulated in the thermal analysis, which can be addressed either using CFD or finite-element method (FEM) code [18]. For structural analysis, FEM codes are adopted to simulate the mechanical response of the structure.

With fire analysis carried out in the fluid domain and thermal and structural analysis performed in the solid domain, numerical simulation involves fluid–structure interaction (FSI), which includes both one-way and two-way couplings. Although the fire–structure interaction is fundamentally two-way, one-way coupling is used in most studies nowadays. In one-way coupling, in order to connect, or “couple”, the different analysis models and boundary conditions must at some point be transferred

between the CFD and FEM modules. To couple the fire and thermal analyses, gas temperatures, incident radiation heat fluxes, convective coefficients, and so forth are transferred from the fire analysis (gas-phase) to the thermal analysis (solid-phase). To couple the thermal and structural analyses, temperature distributions within the structural elements are transferred from the thermal response calculation to structural analysis [19–22]. However, owing to the different mesh sizes and time steps of CFD and FEM, this process is complex and often compromised by its inherent imperfections. For this reason, a new variable called adiabatic surface temperature (AST) was proposed by Wickström et al. [23,24], which is capable of describing complex convective and radiative conditions as one single scalar quantity. In this approach, the Fire Dynamics Simulator (FDS) [25] developed by NIST is commonly used for fire analysis, while thermal and structural analyses are carried out sequentially by adopting the finite element method using ANSYS [26], ABAQUS [27], etc. As shown in Fig. 1, using the single-variable AST as well as the convective heat-transfer coefficient (h), the total heat flux, which accounts for both convective and radiative heat fluxes in fire analysis, can be correctly transferred to the FEM code, where thermal and structural analyses are performed subsequently. With this methodology, some scholars [28–38] conducted a performance-based analysis of structures under fire conditions. For example, Silva et al. [33] provided a fire–thermal–mechanical interface (FTMI) for evaluating the fire–thermal–mechanical behavior of I-shaped columns under localized fires. Polish scholar Glema [38] investigated an approach for assessing the performance of a concrete filled tubular column in an open car-park fire. However, 1D heat conduction is assumed for solid-phase calculations in FDS software, making it difficult to accurately evaluate temperature distributions in solids. Hence, the aforementioned study

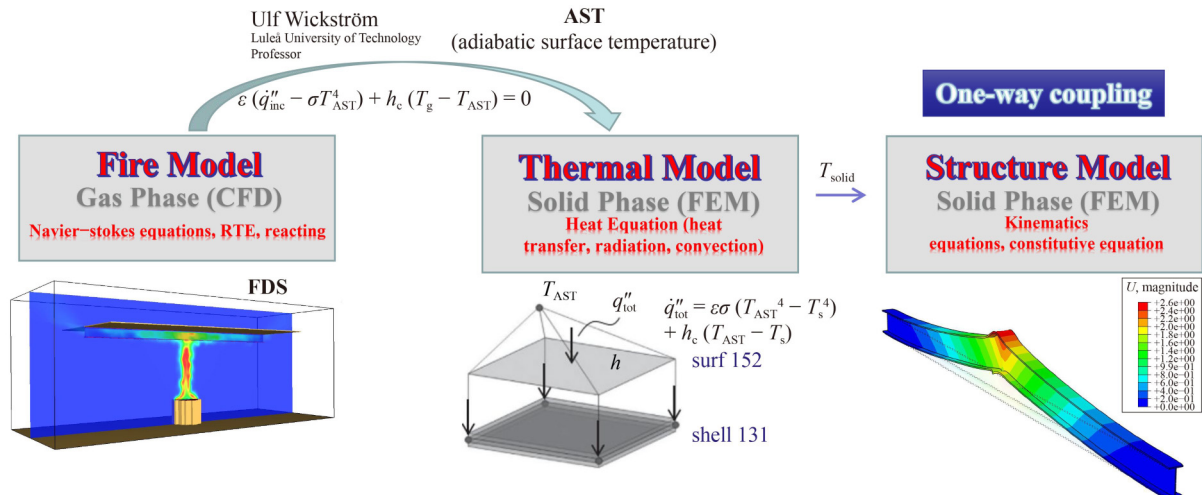


Fig. 1 Mainstream coupled CFD–FEM simulation approach for structural–fire analysis.

could only discard the solid temperature information obtained using the CFD method and apply the AST concept to import the fluid-field information into the FEM analysis module to re-analyze the solid-domain heat transfer. Nonetheless, in actual fire scenarios, the temperatures of the fluid domain and solid domain interact with each other in the process of heat transfer, which is a deficiency of current mainstream CFD–FEM analysis.

The localized fire mostly takes place in open fields where medium or low-speed incoming flows are naturally present (e.g., large stadiums, large car parks). Also, the chimney effect in high-rise buildings owing to the presence of lift shafts can create air movement within the building. Combustion in localized fire scenarios in a building usually involves a buoyancy-controlled turbulent diffusion flame, where the incoming flow introduces additional inertial forces to counteract the buoyancy, complicating the geometry of the flame; therefore, wind fields have a non-negligible effect on fires, motivating many scholars to study the effect. In terms of experimental work, Hu et al. [39–46] carried out a large number of experiments to investigate the effect of cross-air flow on pool fires and proposed that flame-tilting properties are related to the competition between buoyancy and momentum in cross airflows. Chen et al. [47] studied the wind effect on compartment fire under cross-ventilation conditions through experiments. Huang et al. [48] conducted detailed fire-tunnel experiments in a reduced-scale compartment in order to clarify the fire growth process in compartments under external wind conditions. In terms of numerical simulations, Zhou et al. [49] calculated the effects of the longitudinal ventilation velocity of the tunnel on the heat-release rate of high-speed train fires in railway tunnels. Yi et al. [50] studied the influence of canyon crosswind on the burning and flame characteristics of ethanol pool-fires inside a tunnel by FDS. Huang et al. [51,52] conducted an urban fire simulation, adding the consideration of inflow wind velocity to predict the scattering of firebrands. Bridge-deck fire processes with different wind effects and different positions of fire loads were simulated, and the time-dependent temperature laws were obtained using FDS software in some research studies [53,54]. However, all the aforementioned studies focused on the influence of approaching flows on the fire field, and the literature still lacks thermomechanical analyses of structural components subjected to fire under the influence of incoming flows. Therefore, it is necessary to consider the influence of approaching flows to the structure under fire conditions.

In view of the above two aspects of fire–structure coupling analysis and wind–fire coupling analysis, no research has been found that considers the wind-field, fire, thermal, and structural analyses simultaneously. For this reason, this study examines the localized fire scenario

of a steel beam under the ceiling jet affected by different inflow conditions via numerical simulations. By means of CFD–FEM simulation, the coupling analysis of wind, fire, thermal, and structural models is carried out. The correctness of the numerical model is verified by comparing it with the experimental results in the no-flow scenario, and distributions of the velocity and temperature fields under the coupling effect of multiple scenarios are subsequently discussed in detail. The thermal-response properties of the steel beam affected by the ceiling jet fire under different incoming flow conditions are explored, including the wall temperature and radiative and convective heat-flow distributions. Finally, the mechanical response of the steel beam is analyzed in the structural model, and the damaging effects of incoming flow on the ceiling-jet fire and building structure are illustrated in this paper.

2 Methodology

The coupled CFD–FEM method employed in this study is shown in Fig. 2. This approach combines the wind, fire, and thermal models based on the CFD simulation method.

In contrast to the aforementioned boundary-condition solution (applying the AST), this study solves the basic fluid-mechanics equations, species-transport equation using the mixture fraction, etc., in the fluid domain, as well as the heat-conduction equation in the solid domain. At the same time, the radiative and convective heat-flux calculations at the fluid–solid interface are added to achieve a two-way coupling between the three models. Then, the temperature information obtained from thermal analysis of the solid is transferred to the structural finite-element-analysis software for structural analysis, which is summarized in Table 1.

The meaning of all symbols in these formulas can be found in Refs. [55,56], which will not be detailed here.

2.1 Gas phase

The wind and fire models focus on the fluid domain, and the equations to be satisfied include the fluid dynamics control equation (i.e., mass-, momentum-, and energy-conservation equation) [55,56], the mixture-fraction transfer equation using a simple probability model describing the non-premixed combustion, and some auxiliary equations (soot-generation equation, radiative transfer equation, etc.). Some key points are described in detail below.

2.1.1 Energy-conservation equation

In the energy-conservation equation above, $\lambda_{\text{eff}} = \lambda_t + \lambda_r$,

CFD-FEM Methodology

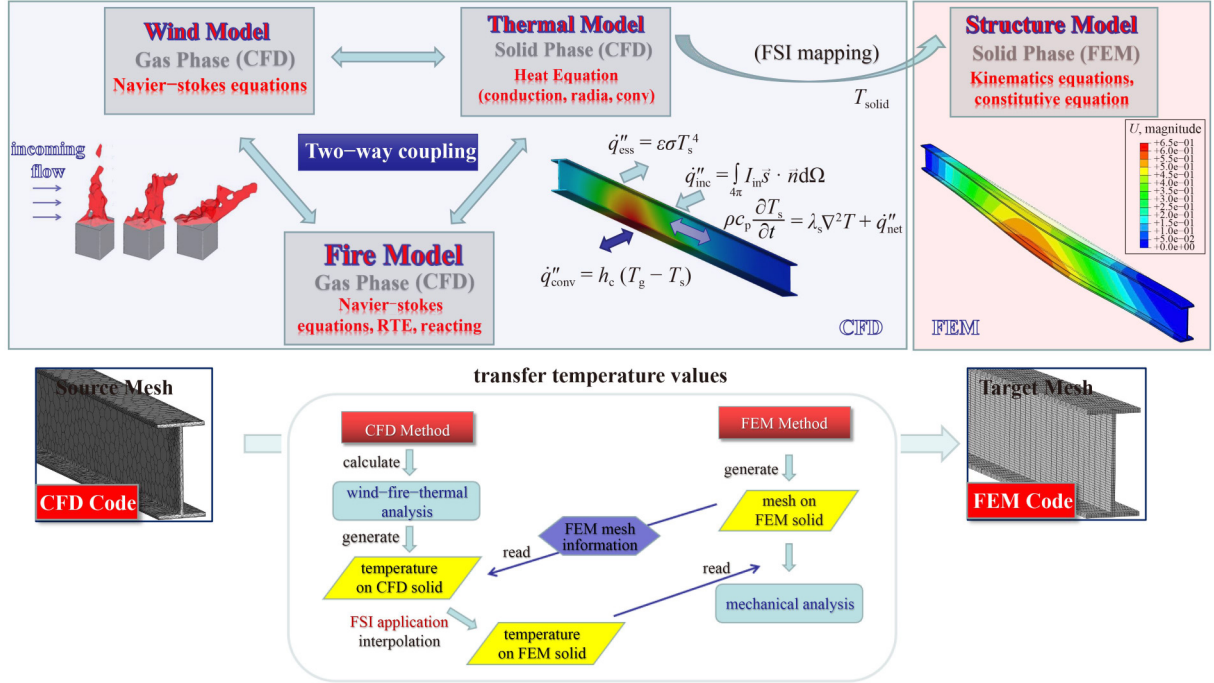


Fig. 2 Two-way coupled CFD–FEM simulation approach for wind–fire–structure analysis.

Table 1 Equations used in CFD–FEM coupling methodology

domain	name of equation	description
gas phase	mass-conservation equation	$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V) = 0$
	momentum-conservation equation	$\frac{\partial}{\partial t} (\rho v_i) + \frac{\partial}{\partial x_j} (\rho v_i v_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} [(\mu + \mu_t) (\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i})] + \rho g_i$
	energy-conservation equation	$\frac{\partial}{\partial t} (\rho H) + \nabla \cdot (\rho V H) = \nabla \cdot (\frac{\lambda_{\text{eff}}}{c_p} \nabla H) + S_h$
	species-transport equation	$\frac{\partial}{\partial t} (\rho f) + \nabla \cdot (\rho V f) = \nabla \cdot (\rho D_{\text{eff}} \nabla f)$
	turbulence/chemistry interaction	$\bar{\varphi}_i = \int_0^1 p(f) \varphi_i(f, H) df$
	radiation-transport equation	$(\Omega \cdot \nabla) I(r, \Omega) = -(\alpha + \sigma_s) I(r, \Omega) + \alpha \frac{\sigma T_g^4}{\pi} + \frac{\sigma_s}{4\pi} \int_{\Omega' \in 4\pi} I(r, \Omega') \Phi(\Omega, \Omega') d\Omega'$
	soot-generation equation	$\frac{\partial}{\partial t} (\rho Y_{\text{soot}}) + \nabla \cdot (\rho V Y_{\text{soot}}) = \nabla \cdot (\frac{\mu_t}{\sigma_{\text{soot}}} \nabla Y_{\text{soot}}) + R_{\text{soot,form}} - R_{\text{soot,comb}}$
fluid–solid boundary	radiative and convective heat-transfer	$q_{\text{net}}'' = q_{\text{rad}}'' + q_{\text{conv}}'' = \varepsilon (q_{\text{inc}}'' - \sigma T_s^4) + h_c (T_g - T_s)$
solid phase	heat-conduction equation	$\rho c_p \frac{\partial T_s}{\partial t} = \lambda_s \nabla^2 T + q_{\text{net}}''$
	virtual work principle	$\int_V \sigma_{ij} \delta \varepsilon_{ij} dV - (\int_V \bar{F}_i \delta u_i dV + \int_{S_p} \bar{T}_i \delta u_i dS_p) = 0$
	material constitutive equation	$\sigma_{ij} = E \varepsilon_{sT} = E (\varepsilon_{ij} - \varepsilon_{ij}^0)$

where λ_{eff} is the fluid effective thermal conductivity and $\lambda_t = C_p \mu_t / \text{Pr}_t$ is the turbulent thermal conductivity, defined according to the turbulence model being used. In this formula, c_p is the thermal capacity, μ_t is the turbulent viscosity, Pr_t is the turbulent Prandtl number, which is generally taken to be 0.85.

H is the total absolute enthalpy of the mixture, expressed as follows.

$$H = \sum Y_i (h_{f,i}^0 + \int_{T_{\text{ref}}}^T c_{p,i} dT), \quad (1)$$

where Y_i is the mass fraction of species i , $h_{f,i}^0$ is the formation enthalpy for species i , and $c_{p,i}$ is the thermal capacity of species i .

In the energy-conservation equation above, two key assumptions are made: first, the Lewis number,

formulated as $L_e = \lambda_{\text{eff}} / \rho c_p D_{\text{eff}}$, is 1; second, Fick's diffusion law can be used to describe the mass flux of the components [57], where D_{eff} is the effective diffusion coefficient, satisfying the formula $D_{\text{eff}} = D_m + \mu_t / \rho S c_t$, and D_m is the mass diffusion coefficient, ρ is air density, $S c_t$ is the turbulent Schmidt number, which is generally taken as 0.7.

2.1.2 Species-transport equation

Turbulent diffused flames, which usually occur in structure fires, is by nature a form of non-premixed combustion [57]. The basis of the non-premixed combustion modeling approach is that under a certain set of simplified assumptions, the instantaneous thermochemical state of the fluid is related to a conserved scalar quantity known as the mixture fraction, f . The major advantage of the application of a mixed fraction lies in the elimination of the source term in the species-transport equation [58].

$$f = \frac{Y_i - Y_{i,\text{ox}}}{Y_{i,\text{fuel}} - Y_{i,\text{ox}}}, \quad (2)$$

where $Y_{i,\text{ox}}$ is the elemental mass fraction of the species i in the oxidant and $Y_{i,\text{fuel}}$ is the elemental mass fraction of the species i in the fuel.

2.1.3 Turbulence/chemical-reaction interaction

By linking the species-transfer equation with the energy conservation equation through the simplified thermodynamic method, the relationship between the dimensionless enthalpy and mixture fraction is established. The thermochemistry calculations are preprocessed and then tabulated for reference. The value of the scalar $\varphi(f)$ (such as temperature, species concentrations, and density) for different mixing fractions can be calculated and stored in reference tables before conducting the CFD simulation. For a single-mixture fraction in an adiabatic system, φ depends solely on the mixture fraction; furthermore, in the case of non-adiabatic systems, the effect of heat loss/gain is parameterized as follows.

$$\varphi_i = \varphi_i(f, H). \quad (3)$$

The relationship between the mean and instantaneous values depends on the interaction between turbulence and chemical reactions, which is considered a probability density function (PDF). Based on the PDF, the time-average predicted values $\bar{\varphi}$ can be calculated as follows (in non-adiabatic systems).

$$\bar{\varphi}_i = \int_0^1 p(f) \varphi_i(f, H) df. \quad (4)$$

The PDF can be assumed to take on the shape of a β -

function [59], which is defined by the mean mixture fraction $\bar{f}$ and its variance $\overline{f'^2}$, as given by Eqs. (5)–(8).

$$f' = f - \bar{f}, \quad (5)$$

$$p(f) = \frac{f^{\alpha-1} (1-f)^{\beta-1}}{\int f^{\alpha-1} (1-f)^{\beta-1} df}, \quad (6)$$

$$\alpha = \bar{f} \left[\frac{\bar{f}(1-\bar{f})}{\overline{f'^2}} - 1 \right], \quad (7)$$

$$\beta = (1-\bar{f}) \left[\frac{\bar{f}(1-\bar{f})}{\overline{f'^2}} - 1 \right]. \quad (8)$$

Using this approach, only two additional transport equations for the mean mixture fraction $\bar{f}$ and its variance $\overline{f'^2}$ have to be solved to predict chemical reactions during gas-phase combustion. In applying the $k-\varepsilon$ RANS (Reynolds-averaged Navier–Stokes) method, the two additional transport equations are as follows.

$$\frac{\partial}{\partial t}(\rho \bar{f}) + \nabla \cdot (\rho V \bar{f}) = \nabla \cdot \left(\frac{\mu_t}{\sigma_t} \nabla \bar{f} \right), \quad (9)$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \overline{f'^2}) + \nabla \cdot (\rho V \overline{f'^2}) &= \nabla \cdot \left(\frac{\mu_t}{\sigma_t} \nabla \overline{f'^2} \right) \\ &+ C_\varepsilon \mu_t (\nabla^2 \bar{f}) - C_d \rho \frac{\varepsilon}{k} \overline{f'^2}. \end{aligned} \quad (10)$$

In these equations, σ_t , C_ε , and C_d are model constants given by Prieler et al. [60].

2.2 Solid phase

The thermal and structural analysis models focus on the solid domain and require the satisfaction of equations such as the heat-transfer, equilibrium, geometric-coordination, and constitutive equations, which are described in detail below.

2.2.1 Thermal analysis model

For the inside of steel components, the Fourier heat-conduction equation is

$$\rho c_p \frac{\partial T_s}{\partial t} = \lambda_s \nabla^2 T + q''_{\text{net}}, \quad (11)$$

where $q''_{\text{net}} = q''_{\text{rad}} + q''_{\text{conv}}$, q''_{net} is net heat flux, q''_{rad} is radiation heat flux and q''_{conv} is convection heat flux. Details are given in Subsection 2.3.

2.2.2 Mechanical analysis model

Under fire conditions, the steel components are subject to a high temperature, where the constitutive relationship is different from that at room temperature. The total strain ε_{ij} at high temperatures is superimposed by several parts, including the free-thermal-expansion strain, ε_{ij}^0 ; the stress-induced strain, ε_{sT} , which complies with Hooke's Law in relation to the thermal stress; and the creep strain, ε_{crs} .

$$\varepsilon_{ij} = \varepsilon_{ij}^0 + \varepsilon_{sT} + \varepsilon_{crs}, \quad (12)$$

$$\varepsilon_{ij}^0 = [\alpha_T \Delta T \quad \alpha_T \Delta T \quad \alpha_T \Delta T \quad 0 \quad 0 \quad 0]^T, \quad (13)$$

where α_T is thermal expansion coefficient. When the creep effect is ignored [61], the constitutive equation at a high temperature can be expressed as follows.

$$\sigma_{ij} = E \varepsilon_{sT} = E(\varepsilon_{ij} - \varepsilon_{ij}^0), \quad (14)$$

where σ_{ij} is stress matrix and E is elasticity modulus matrix.

2.3 Fluid–solid interface

The mechanical response subject to wind loading is very small and can be ignored herein. In this paper, we only discuss the thermal effect between the fluid domain and solid domain. The thermal effect at the interface includes radiative heat transfer and convective heat transfer.

$$q''_{\text{net}} = q''_{\text{rad}} + q''_{\text{conv}}. \quad (15)$$

2.3.1 Radiative heat transfer

The incident radiation heat flux q''_{inc} is calculated by integrating the radiation intensity approaching the solid surface.

$$q''_{\text{inc}} = \int_{4\pi} I_{\text{in}} \cdot \vec{s} \cdot \vec{n} d\Omega, \quad (16)$$

where I_{in} is the intensity of the coming ray. $\vec{s}$ is the ray direction vector and $\vec{n}$ is normal direction pointing out of the domain. Most building materials are opaque and can be treated as gray bodies.

Therefore, net radiative heat flux is calculated by

$$q''_{\text{rad}} = \varepsilon(q''_{\text{inc}} - \sigma T_s^4), \quad (17)$$

where ε is the surface emissivity, T_s is the surface temperature. In this formula, σ is the Stefan-Boltzmann constant ($5.67 \times 10^{-8} \text{ W} \cdot \text{m}^{-2} \cdot ^\circ\text{C}^{-4}$).

2.3.2 Convective heat transfer

The convective heat flux q''_{conv} depends on the difference between the surrounding gas temperature T_g and the solid surface temperature T_s and is usually calculated using Newton's cooling formula.

$$q''_{\text{conv}} = h_c(T_g - T_s). \quad (18)$$

The convective heat transfer coefficient h_c represents the heat-transfer capacity between a fluid and a solid surface.

3 Case study for validation

The study object is a steel beam under the ceiling-jet condition in the case of approaching flow. The experimental data of Yokobayashi et al. [14] was employed in this study for model validation. Figure 3 shows the experimental setup. A circle diffusion burner with a height of 200 mm and diameter of 500 mm was located directly below a steel I-beam (cross section: 150 mm $\times$ 75 mm $\times$ 5 mm $\times$ 6 mm thickness and 3600 mm length). The distance was 600 mm. Propane was used as the fuel, and the heat-release rate (HRR) was 130 kW. The ceiling was placed on top of the I-beam. Following Yokobayashi et al.'s [14] experiment, the incident radiation heat-flux and temperature measurements were conducted on the steel I-beam. Since the incident radiation heat-flux quickly approaches a steady-state, measurements were taken for ten minutes, and the average value was recorded. The wall temperature was measured at the moment when the steel beam was exposed to fire for 30 min.

According to the placement of measuring points in the experiment, as shown in Fig. 4, the three axes of the lower flange, i.e., the web and upper flange of the I-steel, were selected and labeled as lines 1, 2, and 3. Also, for convenience, two sections of the steel beam were selected: the YZ section ($x = 0$) was located in the axial middle of the I-steel, while the XZ section ($y = 5 \text{ mm}$) was adjacent to the web surface, with a distance of 5 mm between the two. Four points were selected on the surface of the steel beam: point O was the origin of the coordinates; points O, A, and B were located on the YZ section; and point C was 0.5 m upwind of point O.

4 Numerical simulation

4.1 Computational fluid dynamics simulation

4.1.1 Computing domain and grid

A wind-tunnel domain with a size of 25 m $\times$ 8 m $\times$ 4 m was built using Ansys ICFM CFD (Fig. 4). The model

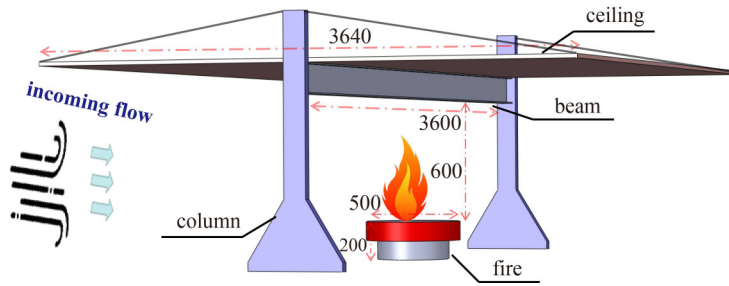


Fig. 3 Experimental model of I-beam subjected to ceiling jet (unit: mm).

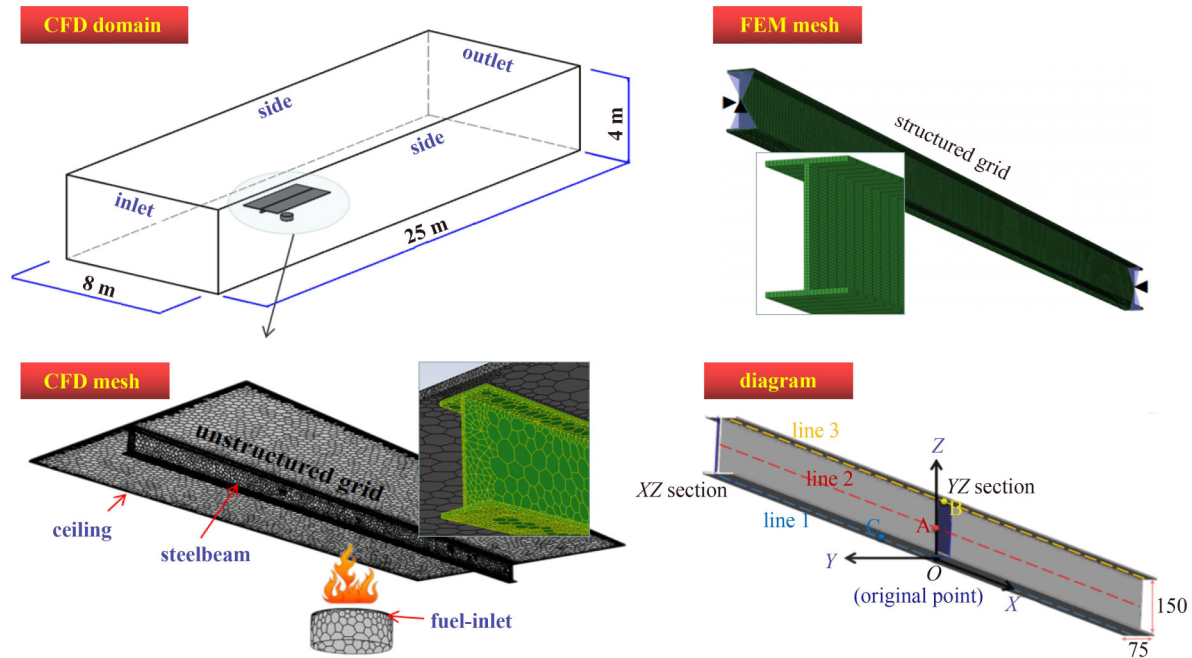


Fig. 4 CFD and FEM numerical models of I-steel subjected to fire.

grid adopted unstructured polyhedral mesh. After repairing, smoothing, and simplifying, the final solid domain was composed of a closed surface with 106524 polyhedron meshes, while the final fluid domain, which lies outside the solid domain, included 450862 meshes. The grid systems are shown in Fig. 4. To calculate the boundary layer around the steel-beam model [62], 30 prism layers were positioned at the surface of the element with a thickness of 0.02 m and growth rate of 1.16 in each cell layer so that the entire surface of the steel beam satisfied $y^+ < 3$.

4.1.2 Numerical method

Ansys Fluent [57] was used to analyze the flow field, heat transfer, and species transport using the finite volume method. Table 2 presents details of the simulation method. Since the fire was time-dependent, the transient, pressure-based solver was used for calculation. The simulation took the effects of gravity into account. The RNG k -turbulence model was applied for the RANS

Table 2 Numerical scheme (model settings)

item	model or status
solver	pressure-based, implicit, transient
turbulence model	RNG k - ε model
radiative heat transfer	DO model
combustion	chemical-equilibrium non-premixed combustion
soot formation	one-step Khan and Greeves model

equations, with the effect of full buoyancy considered. Standard wall functions were used to incorporate the near-wall flow. Thermal radiation, convection, and conduction were considered for heat transfer, and the discrete ordinates radiation model [63] was adopted to simulate the radiative heat transfer. The combustion process involved the turbulent-diffusion non-premixed combustion of propane and air. The simple PDF method was used to describe the interaction between turbulence and combustion statistically, and the non-premixed equilibrium chemical model was employed for calculation. The one-step Khan and Greeves model [64] was

used to predict the rate of soot formation, which was based on a simple empirical rate.

4.1.3 Boundary conditions, calculation settings, and material properties

Table 3 details the boundary conditions and related parameters. The steel-beam surface was regarded as a two-sided wall with a coupled thermal condition, so the solver could calculate heat transfer directly from the solution in the fluid zone adjacent to the solid zone. The emissivity of the steel-beam surface was set to 0.9 in order to simulate the experimental calibration procedure more accurately. The ceiling and the ground were regarded as adiabatic, i.e., no heat was exchanged with the surrounding environment. The fuel-inlet surface was defined as the mass-flow inlet. The mass-flow rate was calculated by the HRR of the experiment. Uniform wind was introduced through the inlet surface and exhausted through the outlet surface. According to the experimental conditions used in a previous study [14], the wind speed was set to 0 m/s for model verification, and the approaching flow was examined using three airspeeds: 0, 1.0, and 1.5 m/s. The turbulence intensity was set to 5%, and the turbulent viscosity ratio was 10. The outlet condition was set as a free outflow based on the mass-conservation law.

The convection term was integrated using the QUICK difference scheme, and the PISO algorithm was used for pressure–velocity coupling. The least-squares cell-based

algorithm was adopted for gradient discretization. The second-order upwind scheme was used in the momentum, turbulence, and energy equations. The solutions were considered as converged when the residual dropped to 1×10^{-6} for the energy equation and 1×10^{-4} for the other equations. Considering the reasonable value of the Courant number, 0.1 s was set as the time-step size, and hence a total of 18,000 time-steps comprised a 1800 s fire exposure.

As for the density calculations, the air mixture (propane–air) can be regarded as the ideal non-compressible gas. The WSGGM method [66] was employed to calculate the absorption coefficient of the gaseous products generated by the flame. The density of steel was 7850 kg/m^3 ; the thermal properties such as thermal conductivity and specific heat capacity, which affect the elevated temperature of steel, were used as specified in Eurocode 3 [65].

4.2 Finite-element-method simulation

4.2.1 Mesh discretization and boundary conditions

The ABAQUS software package [27] was used for the finite-element modeling and structural analysis of the steel beam. To model the steel-beam element, a three-dimensional (3D), eight-noded hexahedral element with three degrees of freedom in each node (C3D8R) was adopted. A FE model with a mesh size of 25 mm was selected for this study, as shown in Fig. 4. The fixed-

Table 3 Boundary conditions for simulation setting

item	boundary type	boundary condition setting/ simulation setting/material property
steel-beam	two-sided wall	emissivity: 0.9 shear condition: no-slip
inlet	velocity inlet	velocity magnitude: 0, 0.5, 1.0, and 1.5 m/s turbulent intensity: 5% mean mixture fraction: 0
fuel inlet	mass-flow inlet	mass flow rate: 0.00189 kg/s turbulent intensity: 5% mean mixture fraction: 1
ground ceiling	wall	thermal condition: adiabatic wall shear condition: no-slip
outlet	outflow	
side	symmetry	
velocity–pressure coupling		PISO
discretization scheme for momentum, turbulence, and energy equation		second-order upwind scheme
time-step size		0.1 s
flow time		1800 s
mixture material (propane–air)		non-compressible ideal gas
absorption coefficient of flame gas		WSGGM method
thermal properties for steel		following Eurocode 3 [65]

boundary conditions were satisfied by restraining the displacement (UX , UY , and UZ) and rotational degrees (RX , RY , and RZ) of freedom at the beam ends. The upper surface of the upper flange contacted the ceiling and was also considered to be a fixed constraint.

4.2.2 Material properties

To appropriately model the response of the exposed steel member under fire conditions, its mechanical material properties need to be defined; in the FEM analysis model, elevated temperatures are the key parameters. The elastic–isotropic option was selected to model the Q235 steel materials. The yield strength and the plastic strain values were modeled using the plastic–isotropic option. Poisson’s ratio was taken as 0.3 in accordance with EC3 [66]. The T. T. Lie high-temperature stress-strain material model for steel [67] is shown below.

$$\begin{cases} \sigma = E_T \varepsilon, & (\varepsilon \leq \varepsilon_{pT}) \\ \sigma = (12.5\varepsilon + 0.975)f_{yT} - \frac{12.5f_{yT}^2}{E_T}, & (\varepsilon > \varepsilon_{pT}) \end{cases} \quad (19)$$

with

$$\varepsilon_{pT} = \frac{0.975f_{yT} - 12.5f_{yT}^2}{E_T - 12.5f_{yT}}, \quad (20)$$

where

$$\frac{E_T}{E_{20}} = 1.0 + \frac{T}{2000 \ln(T/1100)}, \quad (T \leq 600^\circ\text{C}) \quad (21)$$

$$\frac{E_T}{E_{20}} = \frac{690 - 0.69T}{T - 53.5}, \quad (600^\circ\text{C} < T \leq 1000^\circ\text{C}) \quad (22)$$

and

$$\frac{f_{yT}}{f_{y20}} = 1.0 + \frac{T}{900 \ln(T/1750)}, \quad (T \leq 600^\circ\text{C}) \quad (23)$$

$$\frac{f_{yT}}{f_{y20}} = \frac{340 - 0.34T}{T - 240}, \quad (600^\circ\text{C} < T \leq 1000^\circ\text{C}) \quad (24)$$

The equation for the thermal expansion coefficient α_s recommended by NIST TN1681 [68] is as follows.

$$\alpha_s = 1.17 \times 10^{-5} + 1.34 \times 10^{-8} T_s - 9.7 \times 10^{-12} T_s^2 + 1.67 \times 10^{-16} T_s^3. \quad (25)$$

4.3 Computational fluid dynamics–finite element method coupling

As mentioned in the introduction of this paper, most researchers use FDS for fire analysis, apply the AST

concept to transfer the fire information, and then use FE methods for thermal and structural analyses. In this study, the use of CFD methods allows two-way coupling of fire and thermal analyses, which is a more rational consideration of the heat transfer between the fluid and solid domains. After thermal analysis, only the solid temperature of the component needs to be imported to the FEM module. The difference in modeling the steel beam in CFD and FEM leads to different numerical grids for the steel beam. Therefore, the solid temperature in the CFD module has to be mapped on the numerical grid applied in the FEM simulation. Figure 2 shows the means of operation when information on the internal temperature distribution of the member is transferred between the CFD and FEM analyses.

5 Results and discussion

5.1 Model validation

5.1.1 Grid-independent study

A grid-independent study was conducted for no-incoming-flow conditions to ensure that the numerical results were unaffected by the number of mesh cells in the computational domain. Figure 5 shows the variation of the solid temperature at four points on the surface of the steel beam during the fire; the three symbols indicate the three different mesh models. Mesh 2 (560862 polyhedrons) and Mesh 3 (926732 polyhedrons) were based on Mesh 1 (2728433 triangles) and were transformed into polyhedral grids, resulting in a significant reduction in the number of grids. The degree of curve agreement confirms the irrelevance of the grids to the numerical simulation results. Considering the computation speed and accuracy of the simulation results, the Mesh 2 model was selected for the subsequent numerical study.

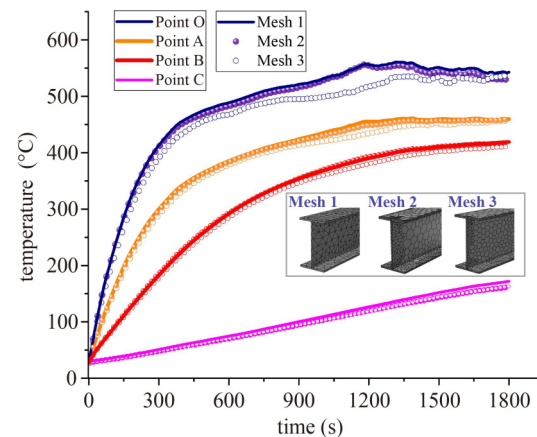


Fig. 5 Surface-point temperature variation.

5.1.2 Comparison with experimental results

Figure 6 shows a 3D diagram of the 400 °C isothermal surface of the flame obtained from the FLUENT simulation. The ceiling jet has a semi-constrained, gravitationally stratified flow; horizontal flow occurs when the flue gas accumulates to a certain thickness under the ceiling. As can be seen in the figure, the impact area of the plume on the ceiling was generally round.

Figure 7 shows the wall temperature at 1800 s when the steel beam was subjected to fire. As the middle region is directly heated by the flame, the temperature here is the highest, with the lower flange reaching close to 600 °C. The web and upper flange have similar temperatures, which are slightly lower than that of the lower flange. Figure 7 shows the axial variation curves of the wall temperatures of lines 1 and 2 (only the half-wall temperature is displayed because the wall temperature is

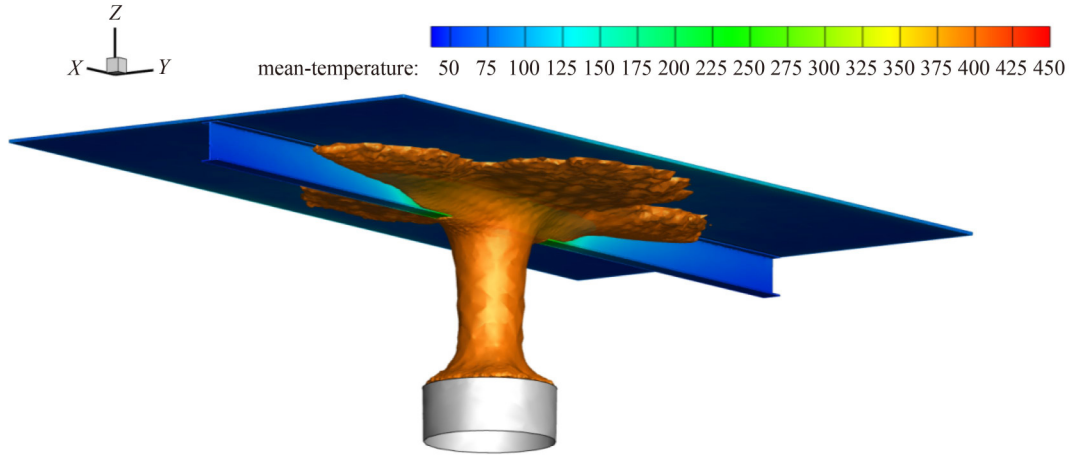


Fig. 6 Isothermal surface (400 °C) of the flame without incoming flow (unit: °C).

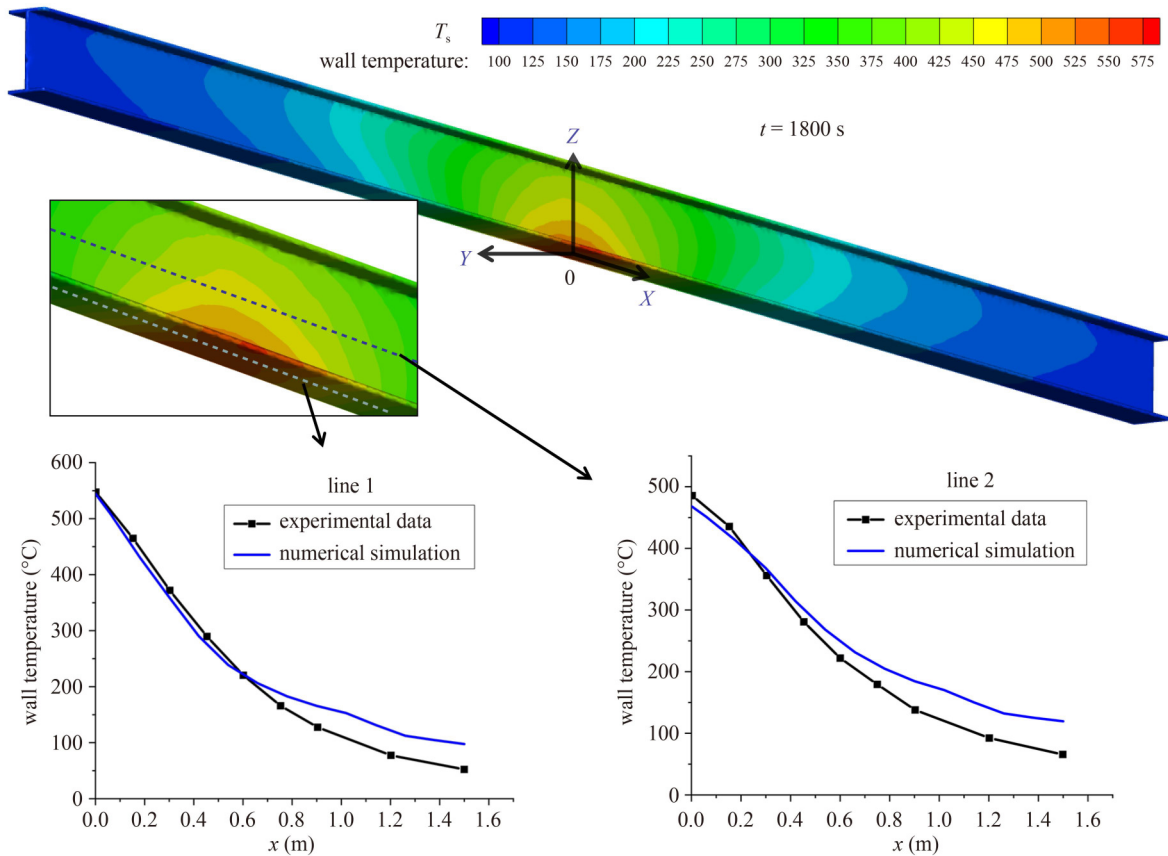


Fig. 7 Temperature distributions of steel beam at 1800 s (unit: °C).

symmetrically distributed). The numerical simulation results in the central high-temperature region are in good agreement with the experimental results. Extending from the middle to both ends, the temperature gradually decreases and produces an asymmetrical distribution, while the numerical simulation results are slightly higher than those of the experiments at the end.

Figure 8 shows the mean surface incident radiation heat-flux $\overline{q''_{inc}}$ of the steel beam (600 s time-averaged results). The symmetric distribution pattern of $\overline{q''_{inc}}$ is similar to that of the temperature; it is high in the middle region of the beam, reaching a maximum of 32 kW/m^2 in the numerical simulation, and decreases rapidly as it extends from the middle to the ends, where it is already below 5 kW/m^2 at $x = 0.5 \text{ m}$. The value of $\overline{q''_{inc}}$ is the highest at the lower flange of the beam, followed by the web, with the lowest value occurring at the upper flange. Figure 8 also shows the $\overline{q''_{inc}}$ curves along the axial directions of lines 1 and 2. The numerical simulation results are in good agreement with the experimental results of Yokobayashi et al. [14]. The simulation results are slightly lower than the experimental results for the middle of the I-beam and are closer to the experimental results at the end of the beam. The accuracy of the numerical simulations was verified by comparison with the test results.

5.2 Wind and fire analyses

After the accuracy of the numerical model was verified, the wind model was activated to test the impact of incoming flow on the fire-affected steel beam. For comparison purposes, the partial results of the windless conditions are also given.

5.2.1 Speed field

Figure 9 shows the z -direction velocity distribution of gas in the YZ section in a windless fire scenario obtained by numerical simulation, where the arrow size represents the relative magnitude of the combined velocity and points in the direction of the combined velocity vector. As can be seen, the updraft flow changes from vertical to horizontal when it encounters the ceiling, spreading along the underside of the ceiling in all directions. The updraft reaches a maximum speed of nearly 3 m/s , with the highest velocity occurring directly below the lower flange and on both sides. In addition, after it encounters the lower flange, the updraft generates shunting, with vortices formed on both sides, resulting in a local area with low wind speeds. As there is no incoming flow, the flue-gas flow is distributed symmetrically.

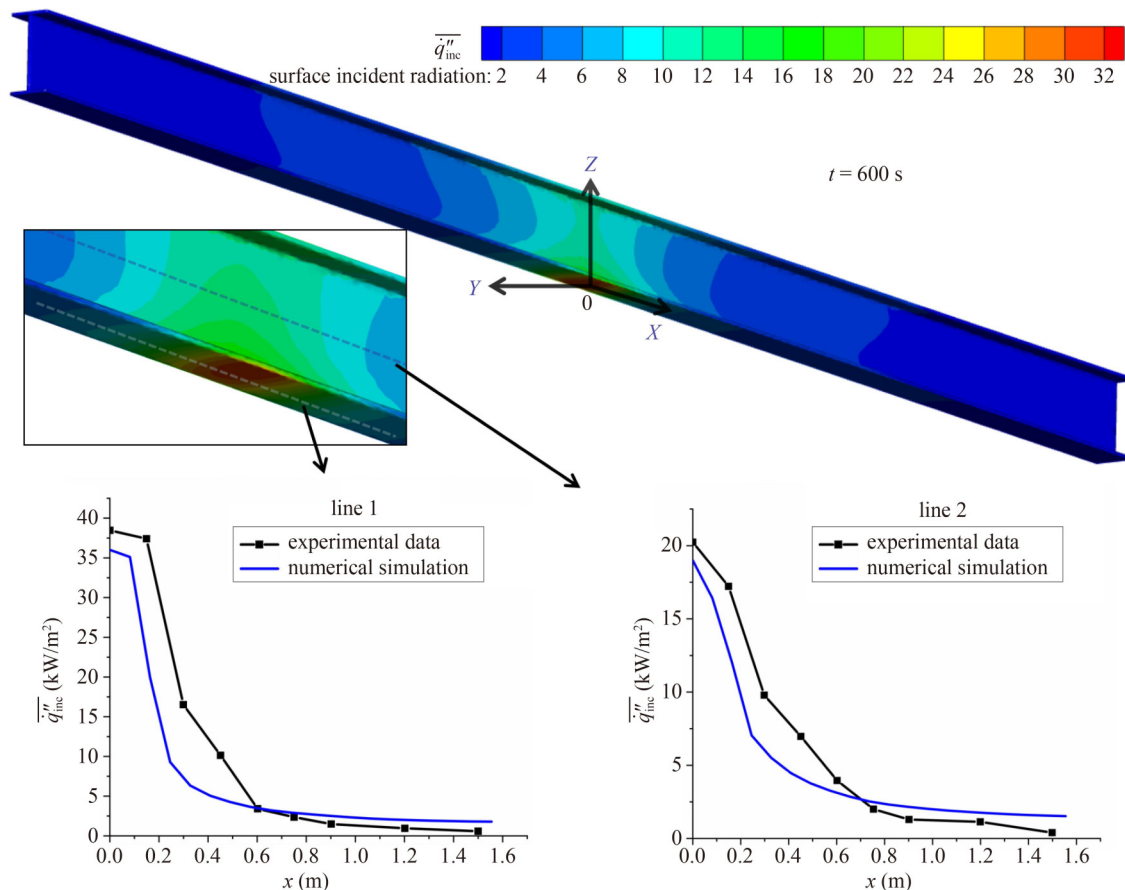


Fig. 8 Surface incident radiation heat-flux distributions of the steel beam in kW/m^2 .

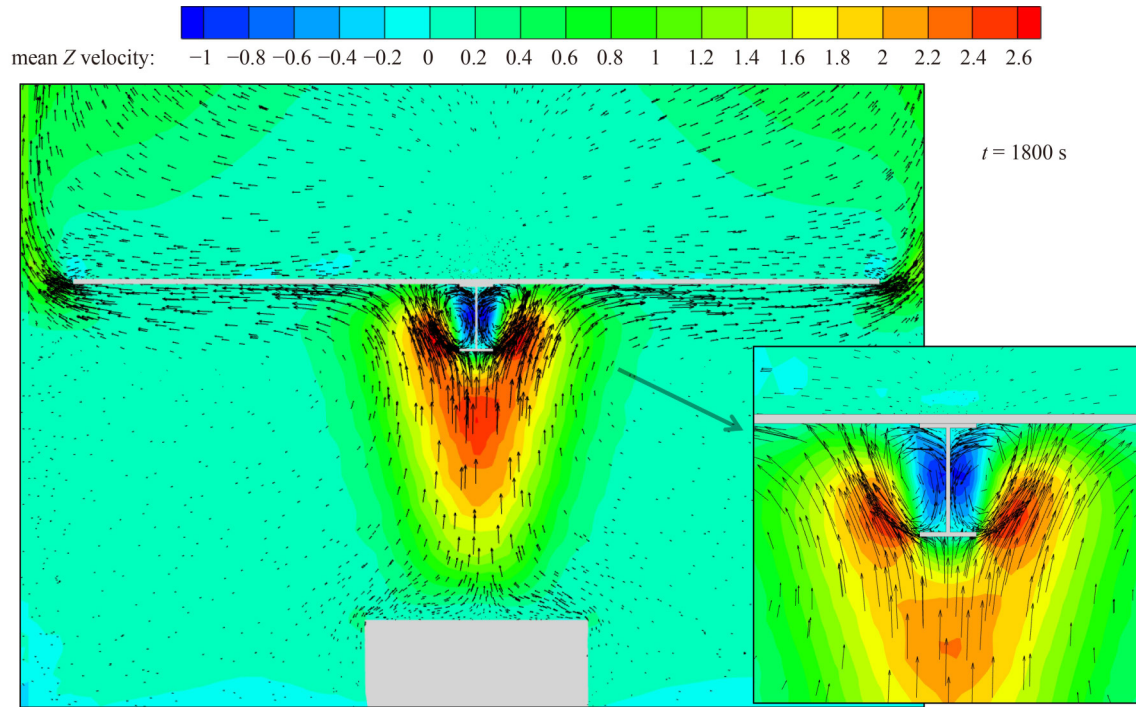


Fig. 9 Velocity distribution in the z -direction without flow (1800 s time-averaged result; unit: m/s).

Figure 10 shows the x -direction velocity distribution of the XZ cross-section without incoming flow and with 1 m/s incoming flows, respectively. As can be seen from the graphs, there is a noticeable tilting of flue gas under the influence of incoming flow, which becomes more significant with increasing flow speed. Meanwhile, the position at which the high-temperature flue gas impinges on the lower flange of the I-beam moves toward the leeward end. The high-temperature flue gas moves obliquely upwards from the fire source, gathering in the local area formed by the upper and lower flanges of the I-steel to produce vortices (shown in Fig. 9) that spiral forward along the direction of flow.

5.2.2 Temperature field

Figure 11(a) shows the gas-temperature distribution in the YZ section obtained by numerical simulation. Owing to the existence of I-steel, the spatial temperature field of fire is very different from the ceiling jet flow without the steel beam. Some of the fuel and oxygen converge on both sides of the web, and their chemical reaction results in the high-temperature region in the upper and lower flanges of the I-steel, with the highest temperature reaching $\approx 1150^\circ\text{C}$ (red zone in Fig. 11(a)). This can be explained by the mixture fraction, as shown in Fig. 11(b), which is ≈ 0.1 in this region, close to the equivalent ratio of propane combustion where the fuel reacts most fully with oxygen and thus emits the highest amount of heat.

Considering the effect of flow, Fig. 12 displays a 3D scheme of the 200°C isothermal surface of the fire plume

at an incoming flow velocity of 0.5 m/s. As the figure suggests, the incoming flow influences the plume morphology to a great extent, which is eventually balanced by the lateral inertia of the incoming flow and the upwards buoyancy of the high-temperature plume.

Figures 10(a) and 10(b) illustrate the gas-temperature distribution in the XZ section without incoming flow and with 1 m/s incoming flows, respectively. The spatial temperature field distribution of fire changed considerably in response to the incoming flow. In the absence of incoming flow, the temperature distribution is symmetric. By contrast, the flame was significantly blown askew under the influence of incoming flow. The highest gas temperature occurs in the leeward area of the brazier, close to its surface. The temperature of the hot flue gas in contact with the I-steel is much lower than that without incoming flow, and it drops dramatically with increasing wind speed.

5.3 Thermal analysis

5.3.1 Wall temperature

The left side of Fig. 13 shows the wall temperatures of the I-beam at 1800 s, when the incoming flow is considered; the curves of the wall temperature along the axial direction of lines 1, 2, and 3 are shown on the right. In the axial direction, the temperatures of the steel beams are no longer symmetrically distributed, but two temperature peaks successively appear on one side, forming “humps” in the distribution. As the flow-velocity

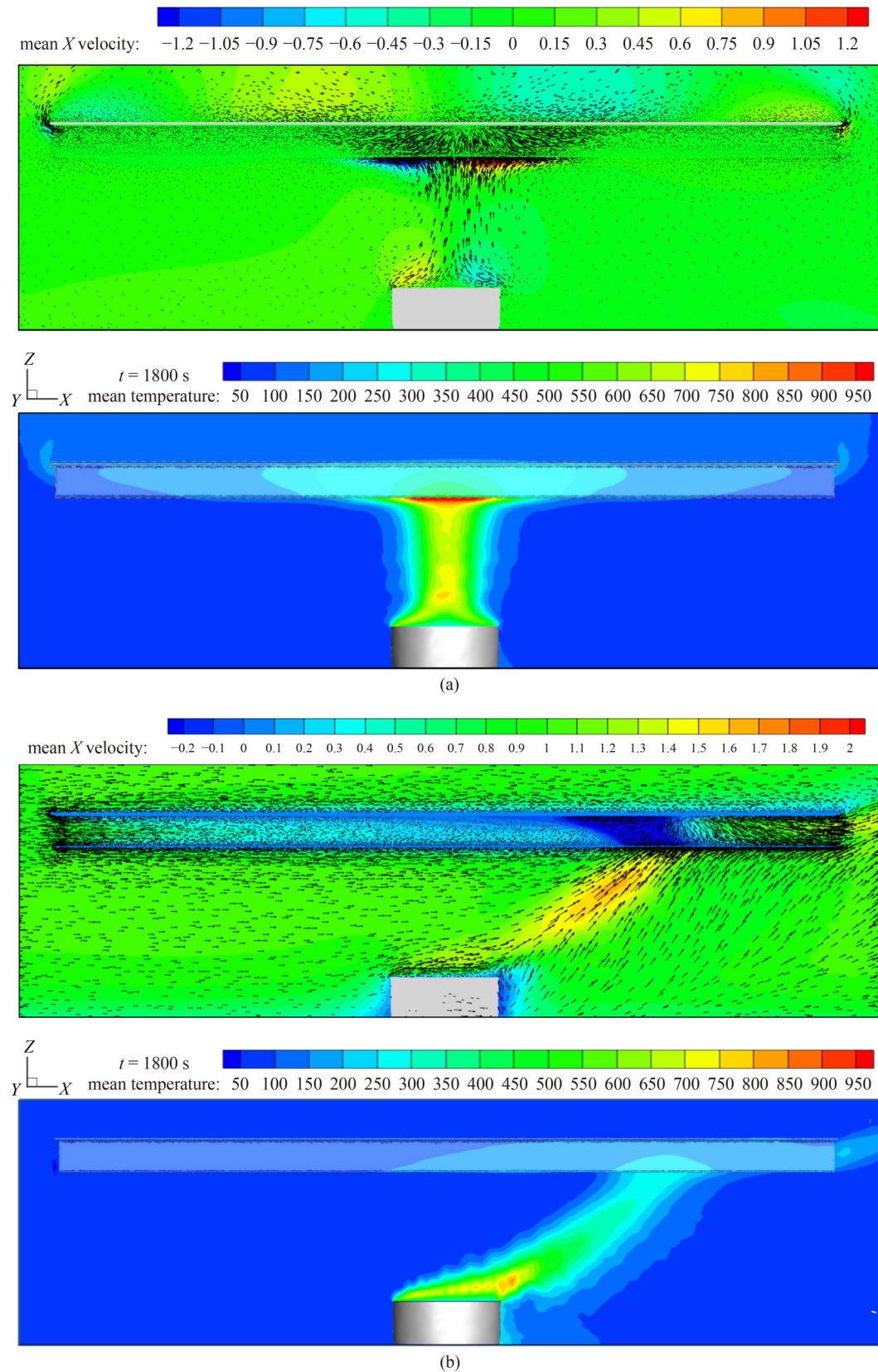


Fig. 10 Velocity (m/s) and temperature ($^{\circ}\text{C}$) distribution generated using the 1800 s time-averaged results: (a) without flow; (b) wind speed of 1.0 m/s.

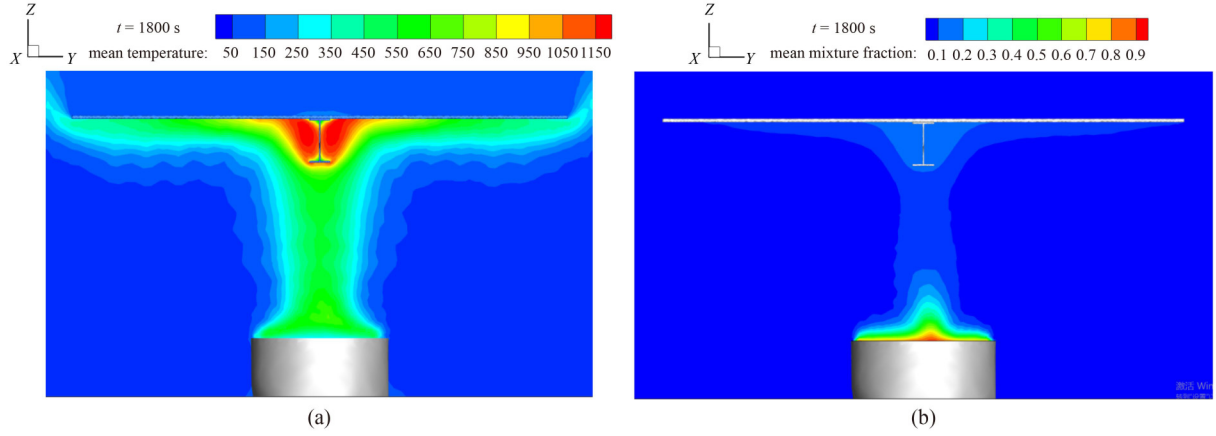


Fig. 11 Temperature and mixture-fraction distribution without cross flow (1800 s time-averaged results): (a) temperature distribution (°C); (b) mixture-fraction distribution.

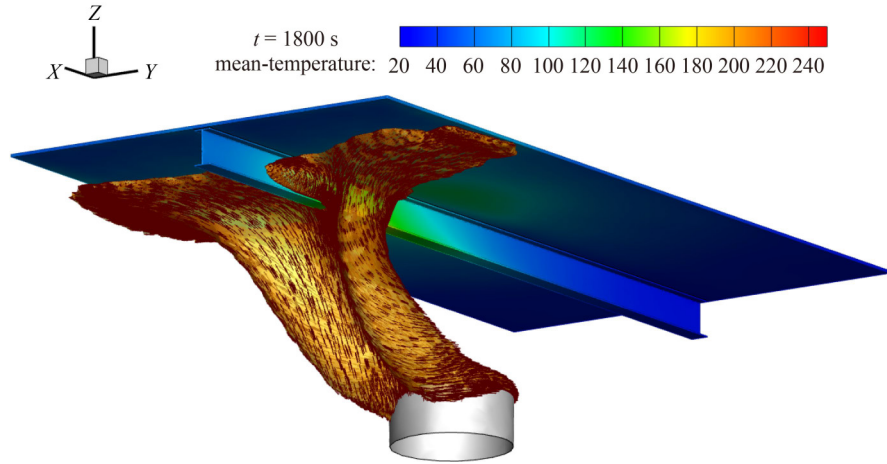


Fig. 12 Isothermal surface (200 °C) of flame with 0.5 m/s incoming flow.

increases, the distance between the two temperature peaks grows. The lower flange temperature of the steel beam is the highest, followed consecutively by that of the web and the upper flange temperature. In Fig. 7, the wall temperature of the I-beam without incoming flow is basically axisymmetric, with only one peak in the proximity of $x = 0$ that reaches a maximum of nearly 600 °C. On the other hand, the maximum wall temperature of the I-steel is ≈ 250 °C when the 0.5 m/s flow is considered, and the maximum temperature is only ≈ 200 °C when the wind speed increases to 1.5 m/s. It can be seen that in this particular case, the heating of the I-steel is remarkably weakened when subject to the influence of incoming flow.

5.3.2 Wall heat-flux

Next, the heat flux received by the I-beam in a fire environment is analyzed, which includes radiative heat flux and convective heat flux. The left and right of Fig. 14 show the mean net radiation heat-flux $\overline{q''_{\text{rad}}}$ and

convective heat-flux $\overline{q''_{\text{conv}}}$ distributions for the combustion process, respectively. Figure 15 is the axial variation curve of radiative and convective heat fluxes received by the lower flange (line 1), web (line 2) and upper flange (line 3).

The gas temperature around the I-beam is higher than the wall temperature, and hence the $\overline{q''_{\text{conv}}}$ of the whole beam is positive; the higher these values, the closer they are to $x = 0$, reaching a maximum of ≈ 15 kW/m² at the lower flange. The convective heat-flux curve suggests significant fluctuations owing to the flame pulsation in combustion. From a hydrodynamic perspective, flame pulsation is the result of the density difference under gravity between the rising flame plume and the surrounding air, which is called Rayleigh–Taylor instability.

The distribution of the net radiative heat flux $\overline{q''_{\text{rad}}}$ is more complex than that of convection. For the lower flange, the incident radiation heat-flux $\overline{q''_{\text{inc}}}$ near $x = 0$ directly exposed to the fire is very large—higher than the radiation flux released by the lower flange itself. This leads to the positive $\overline{q''_{\text{rad}}}$ in this range and the maximum

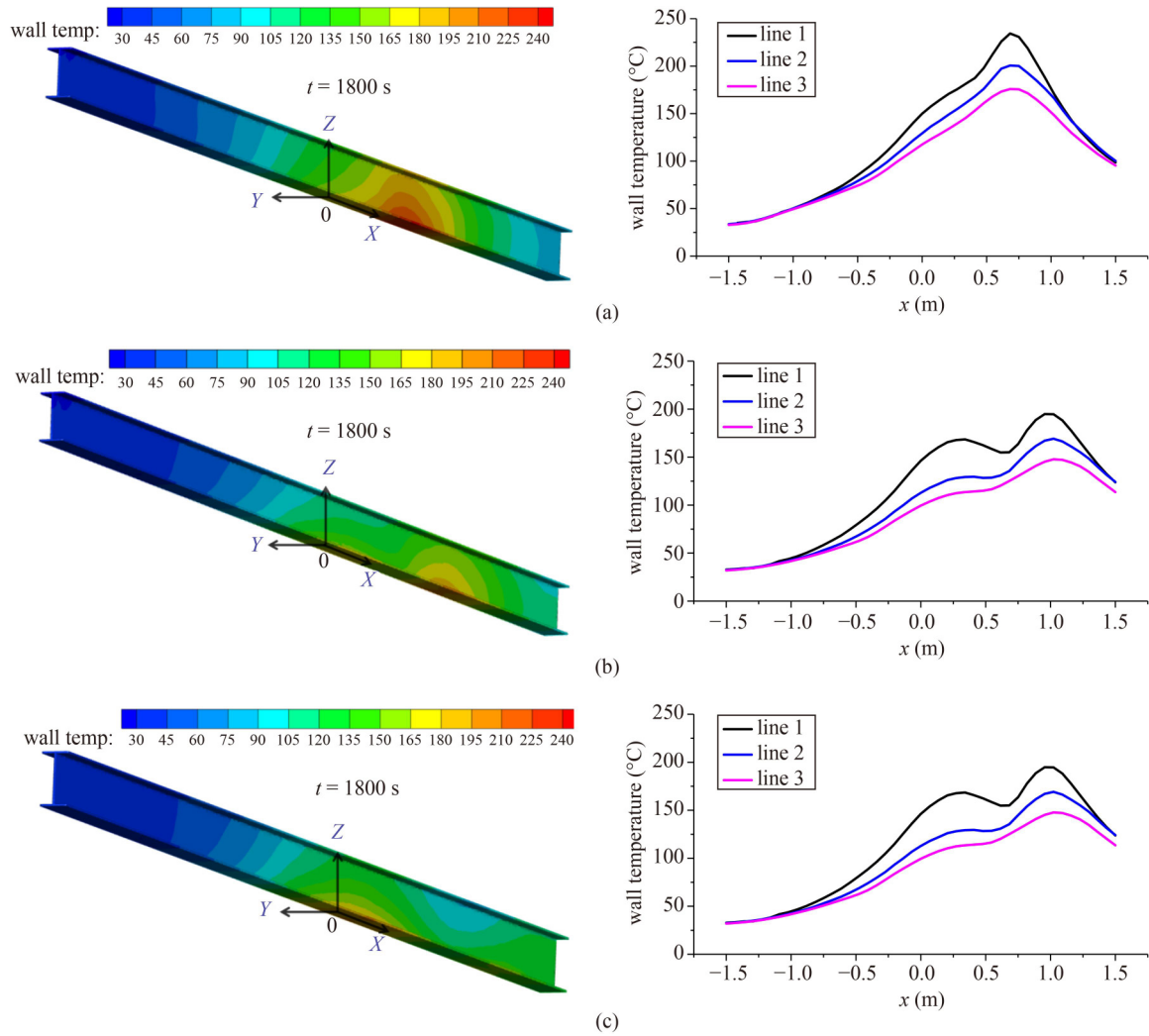


Fig. 13 Wall-temperature distribution with cross flow (1800 s moment results, unit: °C): (a) wind speed of 0.5 m/s; (b) wind speed of 1.0 m/s; (c) wind speed of 1.5 m/s.

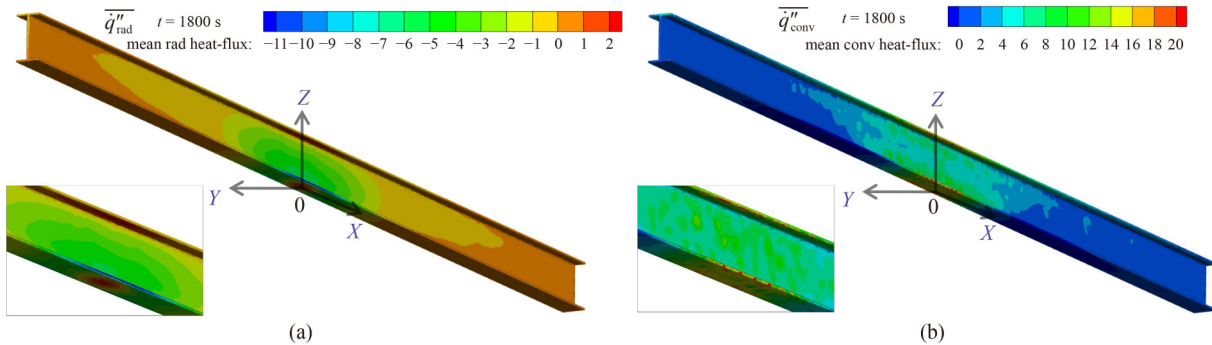


Fig. 14 Wall heat-flux distribution without cross flow (1800 s time-averaged results, unit: kW/m²): (a) mean net radiation heat-flux and (b) convective heat-flux distributions.

value of ≈ 2.5 kW/m². For the adjacent regions on both sides, as shown in Fig. 8, the $\overline{q''_{inc}}$ received decreases rapidly, and the decrease rate of the wall temperature is relatively slow owing to the heat-conduction effect inside the steel, quickly making the $\overline{q''_{rad}}$ become negative, reaching a minimum value of ≈ -4 kW/m² at a distance of

0.2 m from the middle on both sides. It then gradually decreases to 0 along the axial direction toward the ends, indicating that the heat radiation gradually disappears. Owing to the special shape of I-steel, the view factor between the web and flame radiation is very small, resulting in the $\overline{q''_{inc}}$ received by the web being less than

the emission radiation flux released outwards from its surface. This leads to a negative $\overline{q''_{\text{rad}}}$; the closer this value is to $x = 0$, the smaller it is, reaching a minimum of $\approx -4.5 \text{ kW/m}^2$. For the upper flange, $\overline{q''_{\text{rad}}}$ is positive, reaching maximum values of $\approx 2.5 \text{ kW/m}^2$ at $x = 0$. Both radiative and convective heat transfer lead to the temperature rise of the I-beam. Statistically, convective heat transfer contributes more to this temperature rise than radiative heat transfer.

Figure 16 displays the distribution curves of radiative heat flux $\overline{q''_{\text{rad}}}$ and convective heat flux $\overline{q''_{\text{conv}}}$ in different incoming flow conditions and different regions of the I-steel, which can be summarized as follows.

The radiation distributions in the upper and lower flanges (lines 1 and 3) are similar, with the peak value appearing at $\approx x = 0.3 \text{ m}$ with basically no change in the incoming flow velocity. The convection peak constantly shifts downwind and decreases with increasing incoming flow velocity. Taking the lower flange as an example, the peak value of convection decreases from 4 kW/m^2 at 0.5 m/s to 3 kW/m^2 at 1.5 m/s .

For the web (line 2), the radiation effect is significantly reduced, showing a positive–negative distribution along the x -direction. The dividing point moves in the downstream direction with the increase in inflow

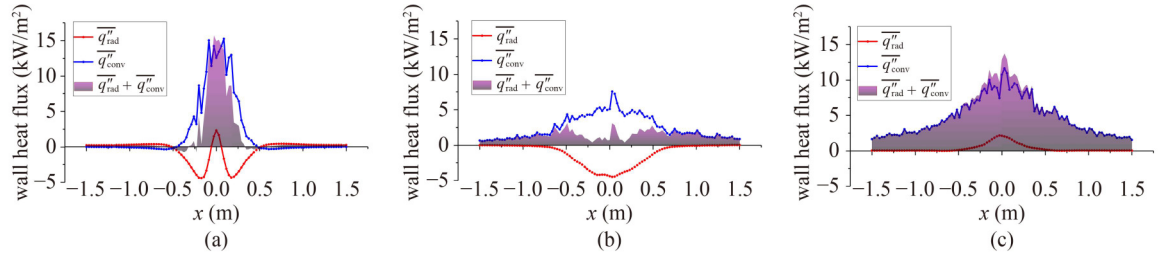


Fig. 15 I-beam wall heat transfer (radiative, convective, and net heat fluxes) without cross flow: (a) line 1; (b) line 2; (c) line 3.

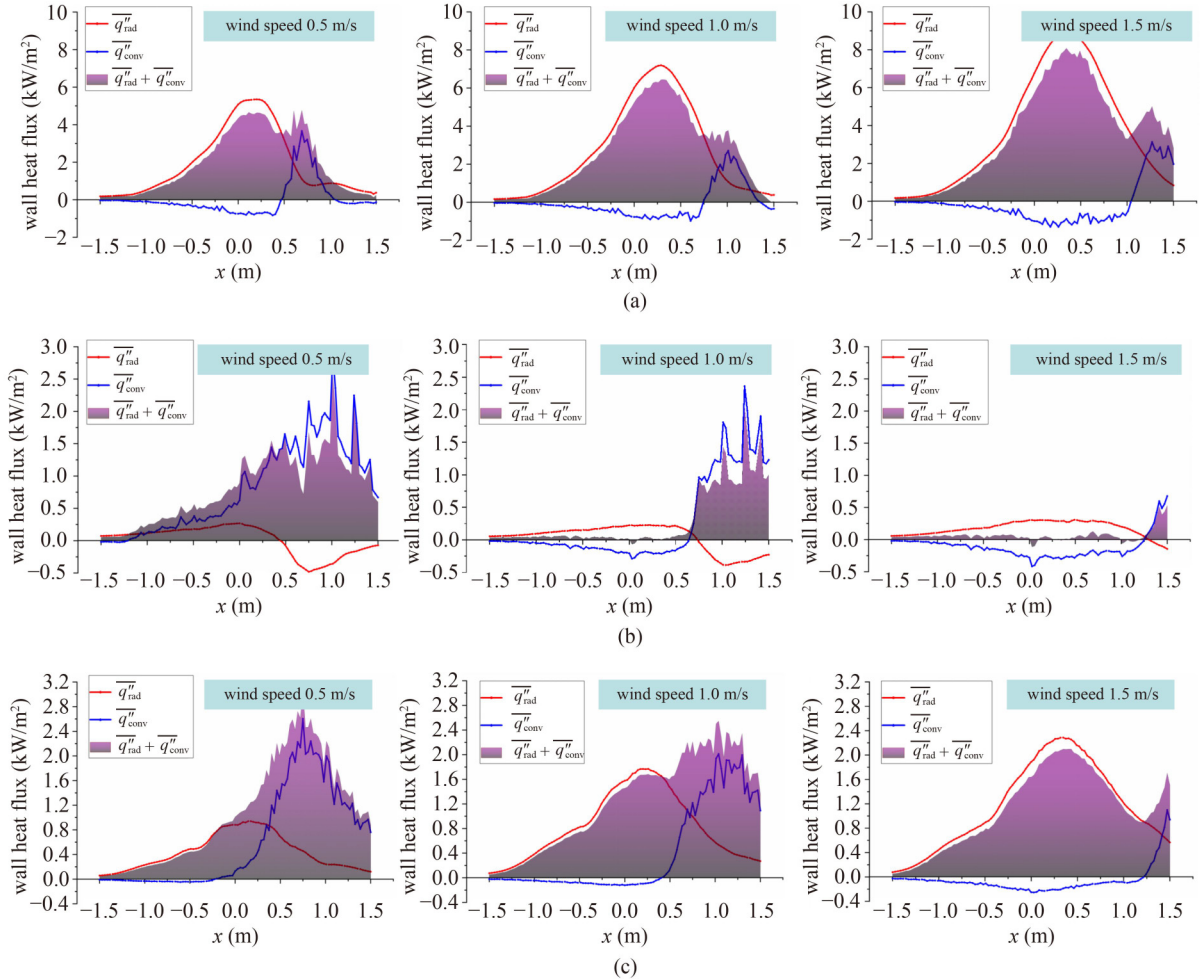


Fig. 16 I-beam wall heat transfer (radiative, convective, and net heat flux) with cross flow: (a) line 1; (b) line 2; (c) line 3.

velocity, and the absolute value of the positive and negative is less than 0.5 kW/m^2 . The convection distribution is similar to that of the flanges, and the peak value shifts along the downwind direction with the increase in flow velocity.

It can also be seen that radiation is dominant in the receiving heat flux of the lower flange, while convection is dominant in that of the web plate. For the upper flange, convection is dominant at low incoming velocity (0.5 m/s), convection and radiation are equivalent at medium velocity (1.0 m/s), and radiation is dominant at high velocity (1.5 m/s). For the I-beam, radiation mainly acts on the $x = -0.25$ – 0.5 m axial zone on the downwind side, while convection mainly acts on the $x = 0.75$ – 1.5 m axial zone in the downwind end.

It can be inferred from the above analysis of radiation and convection of the I-beam that the two wall-temperature peaks (Fig. 13) under the influence of incoming flow are mainly caused by radiative heating and convective heating. The distance between the two temperature peaks grows with the increase in the left incoming flow velocity because the second peak-temperature region dominated by convective heating continuously shifts toward the downstream direction.

5.4 Structural analysis

Figure 17 illustrates the mechanical responses of the steel beam at 1800 s under different incoming flow conditions, which include strain, stress, and displacement responses.

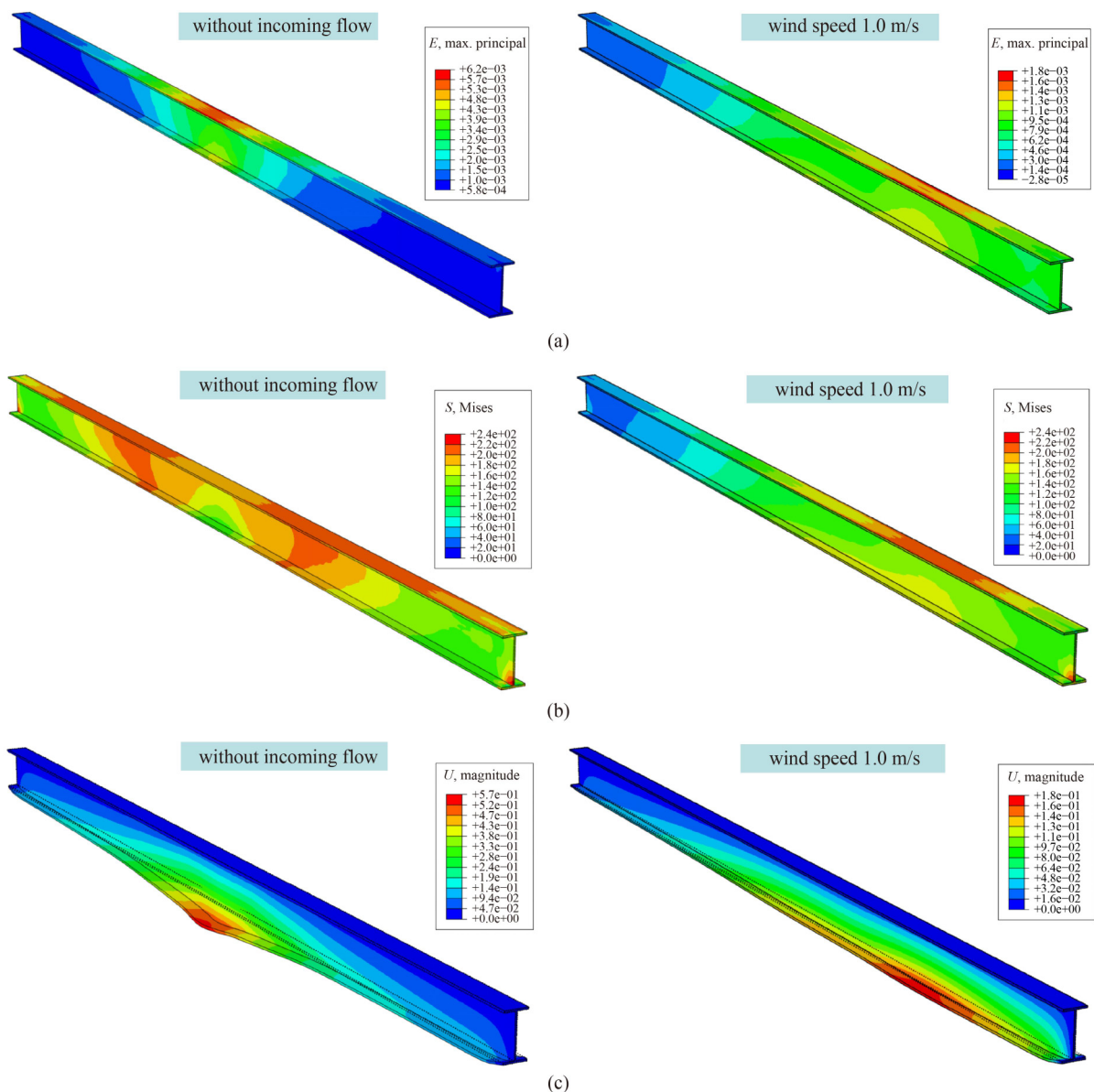


Fig. 17 Mechanical response of steel beam under different incoming flows: (a) strain distribution; (b) stress distribution (unit: MPa); (c) deformation distribution (unit: cm).

The strain distribution of the steel beam in the absence of wind differs from the temperature distribution, with the highest strain occurring in the upper flange region, followed by the web and lower flange. This is because the top surface of the upper flange is considered to be fully restrained. As it extends toward the end of the sides along the x -axis, the temperature gradually drops, and the strain decreases accordingly. Meanwhile, the stress distribution is more complex. At only ≈ 150 MPa, the von Mises stresses in the hottest region of the web and the lower flange are not among the highest values. However, the maximum stress can reach nearly 240 MPa in the proximity of $x = \pm 0.6$ m. This is because, according to the constitutive relationship at high temperatures (Eq. (23) above), the maximum temperature of the beam exceeds 550 °C, and the yield-strength degradation at high temperatures is already evident. Although the strain in the highest-temperature region reaches a maximum and plastic deformation is obvious, the stress value decreases as the yield-strength decreases. Extending axially toward the ends, the yield-strength degradation weakens as the temperature drops significantly, and thus the stress value increases as the yield strength increases. Extending further toward the ends, the stress decreases with the reduction in the strain.

In addition, local buckling is a failure mode commonly observed in thin-walled structural steel elements. This phenomenon is mainly related to the width-to-thickness ratio of the plate. The larger this ratio, the more likely local buckling will occur. At high temperature, the elastic modulus and yield stress of the steel are reduced to varying degrees, complicating the buckling analysis. The above study only considers the constraint effect of the ceiling, so it does not involve the problem of local buckling. It is expected that future work will be carried out to solve this problem.

The mechanical response of the steel beam subject to wind is significantly lower, regardless of stress, strain, or displacement response. When the incoming flow is taken into consideration, the maximum temperature of the beam is only ≈ 240 °C. At this time, the yield strength degradation at a high temperature is insignificant, and the reduction in yield strength is negligible. In addition, the vast majority of the beam is in an elastic state that does not reach the yield strength. The strain distribution of the steel beam is therefore very similar to the stress distribution. For the lower flange and the web, the maximum stress is located in the highest-temperature region at ≈ 160 MPa. The stress and strain of the upper flange are higher than those of the web and lower flange because the upper flange is fixed in place.

Without inflow, the maximum deflection of the steel beam occurs in the middle region, and if inflows are considered, it occurs in the area behind the middle section of the downstream direction. It can be seen from the

figure that the displacement response is very small in the absence of an external mechanical load, and the displacement response without incoming flows is greater than that with flows.

For the examples selected in this paper, the additional wind analysis does not exacerbate the mechanical damage response after fire. However, this does not mean the wind model is meaningless. It is conceivable that the wind has such a huge influence on the fire field that many scenarios exist where winds will exacerbate damage to the structure in a fire situation. To further investigate this possibility, further work is expected to be carried out in the future.

6 Conclusions

In this paper, a numerical simulation method for structural fire analysis was proposed based on CFD–FEM coupling considering the impact of wind.

1) To evaluate the FSI (fluid–structure interaction), wind, fire, thermal characteristics, and structural mechanics were simultaneously considered. In contrast to solving the boundary conditions (applying the AST), as described in the Introduction, this study integrated the fire analysis, influence of wind, and thermal analysis using Fluent software by solving the equations simultaneously. This allowed the two-way coupling between the fluid-temperature domain and solid-temperature domain to be realized, which better reflects the actual heat transfer taking place. After the internal temperature distribution of steel was obtained, it was transmitted to FEM for mechanical response analysis.

2) The validity of the numerical simulation method was verified by comparing the results with the experimental results of an I-beam located under the ceiling in the absence of incoming flow. The velocity and temperature differed significantly from the ceiling jet without the I-beam. The fire plumed and smoke swirled in the local area of the upper and lower flanges and webs of the I-steel, forming a local high-temperature zone. The contribution of thermal convection was greater than that of thermal radiation as the temperature increased.

3) The plume was noticeably blown askew under the effect of left inflow, and the highest-temperature area was quite different from that of the windless situation. The surface temperature of the I-steel exhibited two-peak distributions characterized by “humps” along the axial direction. Under the effect of incoming flow, convection and radiation were reflected in different regions of the I-steel in the axial direction. The reason for the bimodal temperature distribution was elucidated: the first peak-temperature region was mainly caused by radiative heat transfer, and the second was mainly due to convective heat transfer.

4) In terms of mechanical response, under windless conditions, the yield strength degraded remarkably in the highest-temperature region. As a result, the maximum stress did not occur in this region but rather on the adjacent sides. The mechanical response of the structure with incoming flows was significantly lower than that without wind, resulting in a basically elastic state with similarly patterned cloud distributions of temperature, strain, and stress. Since the applied load was not considered, the displacement deflection of the steel beam was small, and the deflection was higher in the absence than in the presence of incoming flow.

Acknowledgments The financial supports from the National Natural Science Foundation of China (NSFC) (Grant Nos. 52078380 and 51820105013) and the Top Discipline Plan of Shanghai Universities-Class I (No. 2022-3-YB-18) are gratefully acknowledged. This study was also supported by National Key Research and Development Program of 14th Five-Year Plan of China (Project No: 2022YFC3801904).

References

- Zhang C, Li G Q, Wang Y C. Sensitivity study on using different formulae for calculating the temperature of insulated steel members in natural fires. *Fire Technology*, 2012, 48(2): 343–366
- Pesic D J, Blagojevic M, Zivkovic N V. Simulation of wind-driven dispersion of fire pollutants in a street canyon using FDS. *Environmental Science and Pollution Research International*, 2014, 21(2): 1270–1284
- Stern-Gottfried J, Rein G, Bisby L A, Torero J L. Experimental review of the homogeneous temperature assumption in post-flashover compartment fires. *Fire Safety Journal*, 2010, 45(4): 249–261
- Pignatta e Silva V. Determination of the steel fire protection material thickness by an analytical process—A simple derivation. *Engineering Structures*, 2005, 27(14): 2036–2043
- China Association for Engineering Construction Standardization (CECS). Technical Code for Fire Safety of Steel Structure in Buildings. Beijing: China Planning Press, 2006 (in Chinese)
- EC3-1-2. Eurocode 3: Design of Steel Structures—Part 1–2: General Rules—Structural Fire Design. London: British Standard, 2005
- Rein G, Jahn W, Torero J. Modelling of the growth phase of Dalmarnock Fire Test One. In: *Proceedings of 12th International Conference and Exhibition on Fire and Materials 2011*. Hampshire: Interscience Communications, 2011
- Rackauskaite E, Bonner M, Restuccia F, Anez N F, Christensen E G, Roenner N, Wegrzynski W, Turkowski P, Tofilo P, Heidari M, Kotsovinos P, Vermesi I, Richter F, Hu Y, Jeanneret C, Wadhvani R, Rein G. Fire experiments inside a very large and open-plan compartment: x-TWO. In: *Proceedings of 11th International Conference on Structures in Fire*. Brisbane: The University of Queensland, 2020
- Zhang C, Silva J G, Weinschenk C, Kamikawa D, Hasemi Y. Simulation methodology for coupled fire-structure analysis: Modeling localized fire tests on a steel column. *Fire Technology*, 2016, 52(1): 239–262
- Kamikawa D, Hasemi Y, Wakamatsu T, Kagiya K. Experimental flame heat transfer and surface temperature correlations for a steel column adjacent to and surrounded by a pool fire. In: *Proceedings of 7th International Symposium on Fire Safety Science*. London: International Association for Fire Safety Science, 2007, 989–1000
- Zhang C, Choe L, Gross J, Ramesh S, Bundy M. Engineering approach for designing a thermal test of real-scale steel beam exposed to localized fire. *Fire Technology*, 2017, 53(4): 1535–1554
- Lineham S A, Thomson D, Bartlett A I, Bisby L A, Hadden R M. Structural response of fire-exposed cross-laminated timber beams under sustained loads. *Fire Safety Journal*, 2016, 85(10): 23–34
- Wiesner F, Peters G, Bisby L A, Hadden R. Structural steel columns subjected to localized fires. In: *Proceedings of International Fire Safety Symposium*. Naples: International Fire Safety Symposium, 2017
- Yokobayashi Y, Hasemi Y, Wakamatsu T, Wakamatsu T. Experimental study on the heating mechanism and thermal response of a steel beam under ceiling exposed to localized fires. *Journal of Structural and Construction Engineering*, 1996, 498(498): 169–175
- Alos-Moya J, Paya-Zaforteza I, Hospitaler A, Rinaudo P. Valencia bridge fire tests: Experimental study of a composite bridge under fire. *Journal of Constructional Steel Research*, 2017, 138: 538–554
- Maraveas C, Vrakas A A. Design of concrete tunnel linings for fire safety. *Structural Engineering International*, 2014, 24(3): 319–329
- Maraveas C. Local buckling of steel members under fire conditions: A review. *Fire Technology*, 2019, 55(1): 59–80
- Welch S, Miles S, Kumar S, Lemaire T, Chan A. FIRESTRUC- Integrating advanced three-dimensional modelling methodologies for predicting thermo-mechanical behavior of steel and composite structures subjected to natural fires. *Fire Safety Science*, 2008, 9: 1315–1326
- Prasad K, Baum H R. Coupled fire dynamics and thermal response of complex building structures. *Proceedings of the Combustion Institute*, 2005, 30(2): 2255–2262
- Luo C, Chen L, Lua J, Shi J. A 3D ABAQUS toolkit for thermal-mechanical damage prediction of composite sandwich structures subjected to fire. In: *Proceedings of 50th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*. Reston: American Institute of Aeronautics and Astronautics, 2009, 1–20
- Baum H R. Simulating fire effects on complex building structures. *Mechanics Research Communications*, 2011, 38(1): 1–11
- Chen L, Luo C, Lua J. FDS and Abaqus coupling toolkit for fire simulation and thermal and mass flow prediction. *Fire Safety Science*, 2011, 10: 1465–1477
- Wickström U, Duthinh D, Mcgratan K. Adiabatic surface temperature for calculating heat transfer to fire exposed structures. In: *Proceedings of Interflam 2007 International Interflam Conference 11th Proceedings*. London: Inter Science Communications Limited, 2007
- Wickström U. Methods for Predicting Temperatures in Fire-Exposed Structures, in *SFPE Handbook of Fire Protection*

- Engineering. New York: Springer, 2016, 1102–1130
25. McGrattan K B, Hostikka S, McDermott R, Floyd J, Weinschenk C, Overholt K. Fire Dynamics Simulator Users Guide. 6th ed. Gaithersburg: National Institute of Standards and Technology, 2013
 26. ANSYS. ANSYS User Manual Version 14.0. 2012
 27. SIMULIA/Abaqus. Abaqus User's Manual Version 6.10 edition. 2010
 28. Zhang C, Li G Q, Wang R. Using adiabatic surface temperature for thermal calculation of steel members exposed to localized fires. *International Journal of Steel Structures*, 2013, 13(3): 547–556
 29. Zhang C, Gross J L, McAllister T, Li G Q. Behavior of unrestrained and restrained bare steel columns subjected to localized fire. *Journal of Structural Engineering*. 2015, 141(10): 04014239
 30. Zhang C, Yu H X, Choe L, Gross J, Li G H. Simulating the fire-thermal-structural behavior in a localized fire test on a bare steel beam. *Engineering Structures*, 2018, 163: 61–70
 31. Zhang C, Zhang Z, Li G Q. Simple vs. sophisticated fire models to predict performance of SHS column in localized fire. *Journal of Constructional Steel Research*, 2016, 120: 62–69
 32. Silva J C G, Landesmann A, Ribeiro F L B. Performance-based analysis of cylindrical steel containment Vessels exposed to fire. *Fire Safety Journal*, 2014, 69: 126–135
 33. Silva J C G, Alexandre L, Fernando L B R. Interface model to fire-thermomechanical performance-based analysis of structures under fire conditions. In: *Proceedings of Fire and Evacuation Modeling Technical Conference 2014*. Manhattan: Thunderhead Engineering, 2014
 34. Silva J C G, Landesmann A, Ribeiro F L B. Fire-thermomechanical interface model for performance-based analysis of structures exposed to fire. *Fire Safety Journal*, 2016, 83: 66–78
 35. Liu Z, Silva J C G, Huang Q, Hasemi Y, Huang Y, Guo Z. Coupled CFD–FEM simulation methodology for fire-exposed bridges. *Journal of Bridge Engineering*, 2021, 26(10): 04021074
 36. Feenstra J A, Hofmeyer H, Van Herpen R A P, Mahendran M. Automated two-way coupling of CFD fire simulations to thermomechanical FE analyses at the overall structural level. *Fire Safety Journal*, 2018, 96: 165–175
 37. Malendowski M, Glema A. Development and implementation of coupling method for CFD–FEM analyses of steel structures in natural fire. *Procedia Engineering*, 2017, 172: 692–700
 38. Szymkuc W, Glema A, Malendowski M, Mielcarek A, Smardz P, Poteralski A. Numerical investigation of fire and post-fire performance of CFT columns in an open car park fire. In: *Proceedings of 10th International Conference on Structures in Fire*. Belfast: Ulster University, 2018
 39. Hu L, Liu S, Xu Y, Li D. A wind tunnel experimental study on burning rate enhancement behavior of gasoline pool fires by cross air flow. *Combustion and Flame*, 2011, 158(3): 586–591
 40. Hu L, Zhang X, Delichatsios M A, Wu L, Kuang C. Pool fire flame base drag behavior with cross flow in a sub-atmospheric pressure. *Proceedings of the Combustion Institute*, 2017, 36(2): 3105–3112
 41. Hu L, Liu S, de Ris J L, Wu L. A new mathematical quantification of wind-blown flame tilt angle of hydrocarbon pool fires with a new global correlation model. *Fuel*, 2013, 106: 730–736
 42. Chen Y, Zhang X, Miao Y, Hu L. Effects of cross airflow and burner distance on temperature profile and flame morphology of dual tandem pool fires. *Fuel*, 2022, 317(1): 123220
 43. Hu L, Liu S, Wu L. Flame radiation feedback to fuel surface in medium ethanol and heptane pool fires with cross air flow. *Combustion and Flame*, 2013, 160(2): 295–306
 44. Hu L, Hu K, Ren F, Sun X. Facade flame height ejected from an opening of fire compartment under external wind. *Fire Safety Journal*, 2017, 92: 151–158
 45. Hu L, Hu J, Liu S, Tang W, Zhang X. Evolution of heat feedback in medium pool fires with cross air flow and scaling of mass burning flux by a stagnant layer theory solution. *Proceedings of the Combustion Institute*, 2015, 35(3): 2511–2518
 46. Hu L, Tang F, Wang Q, Qiu Z. Burning characteristics of conduction-controlled rectangular hydrocarbon pool fires in a reduced pressure atmosphere at high altitude in Tibet. *Fuel*, 2013, 111: 298–304
 47. Chen H, Liu N, Chow W. Wind effects on smoke motion and temperature of ventilation-controlled fire in a two-vent compartment. *Building and Environment*, 2009, 44(12): 2521–2526
 48. Huang H, Ooka R, Liu N, Zhang L, Deng Z, Kato S. Experimental study of fire growth in a reduced-scale compartment under different approaching external wind conditions. *Fire Safety Journal*, 2009, 44(3): 311–321
 49. Zhou Y, Wang H, Bi H, Liu X, Gou Q. Heat release rate of high-speed train fire in railway tunnels. *Tunnelling and Underground Space Technology*, 2020, 105: 103563
 50. Yi L, Luan D, Yang L, Chen T, Tao H, Xu Z, Fan C. Flow field and fire characteristics inside a tunnel under the influence of canyon cross wind. *Tunnelling and Underground Space Technology*, 2020, 105: 103575
 51. Huang H, Ooka R, Kato S, Otake H, Hayashi Y. CFD simulation of thermal plume and firebrands scattering in urban fire. *Fire Science & Technology*, 2004, 23(2): 152–163
 52. Huang H, Ooka R, Kato S, Hayashi Y. A numerical study of firebrands scattering in urban fire based on CFD and Firebrands aerodynamics measurements. *Journal of Fire Sciences*, 2016, 25(4): 355–378
 53. Alos-Moya J, Paya-Zaforteza I, Hospitaler A, Loma-Ossorio E. Valencia bridge fire tests: Validation of simplified and advanced numerical approaches to model bridge fire scenarios. *Advances in Engineering Software*, 2019, 128: 55–68
 54. Ma R, Cui C, Ma M, Chen A. Numerical simulation and simplified model of vehicle-induced bridge deck fire in the full-open environment considering wind effect. *Structure and Infrastructure Engineering*, 2020, 17(12): 1–12
 55. Gajjar J. Introduction to theoretical aerodynamics and hydrodynamics. *The Aeronautical Journal*, 2012, 116(1177): 329–330
 56. Kays W M, Crawford M E. Convective Heat and Mass Transfer. 2nd ed. New York: McGraw-Hill, 1980
 57. Peters N. Laminar diffusion flamelet models in non-premixed turbulent combustion. *Progress in Energy and Combustion Science*, 1984, 10(3): 319–339

58. Sivathanu Y R, Faeth G M. Generalized state relationships for scalar properties in Non-Premixed Hydrocarbon/AirFlames. *Combustion and Flame*, 1990, 82(2): 211–230
59. Theory Guide for ANSYS Fluent 12.1. 2009
60. Prieler R, Demuth M, Spoljaric D, Hochenauer C. Evaluation of a steady flamelet approach for use in oxy-fuel combustion. *Fuel*, 2014, 118: 55–68
61. Li G Q, Zhang C. Creep effect on buckling of axially restrained steel columns in real fires. *Journal of Constructional Steel Research*, 2012, 71: 182–188
62. Oh W, Kato S. The effect of airspeed and wind direction on human's thermal conditions and air distribution around the body. *Building and Environment*, 2018, 141: 103–116
63. G D Raithby, E H Chui. A finite-volume method for predicting a radiant heat transfer in enclosures with participating media. *ASME Journal of Heat and Mass Transfer*, 1990, 112(2): 415–423
64. Khan I M, Greeves G. A Method for Calculating the Formation and Combustion of Soot in Diesel Engines. *Heat Transfer in Flames*. Chapter 25. Washington, D.C.: Scripta, 1974
65. Eurocode 3: Design of steel structures—Part 1–2: General rules—Structural fire design. London: British Standard, 2005
66. T F Smith, Z F Shen J N Friedman. Evaluation of coefficients for the weighted sum of gray gases model. *ASME Journal of Heat and Mass Transfer*, 1982, 104(4): 602
67. Lie T T, Macaulay B A. Evaluation of the Fire Resistance of Protected Steel Columns. Internal Report No. 583. 1989
68. Phan L T, Gross J L, McAllister T P. Best Practice Guidelines for Structural Fire Resistance Design of Concrete and Steel Buildings. Gaithersburg: National Institute of Standards and Technology, 2022