Heat Transfer Deterioration in Upward and Downward Pipe Flows of Supercritical n-Decane for Actively Regenerative Cooling

N.B.: This is the ACCEPTED version of this article. The final, published version of the article can be found at: https://doi.org/10.1016/j.ijthermalsci.2021.107066

Yong Li1,2,5, Christos N. Markides3,4,†, Bengt Sunden5, Gongnan Xie2,6*

1 School of Mechanical Engineering, Northwestern Polytechnical University, P.O. Box 552, Xi’an, 710072, China.
2 School of Marine Science and Technology, Northwestern Polytechnical University, P.O. Box 24, Xi’an, 710072, China.
3 Clean Energy Processes (CEP) Laboratory, Department of Chemical Engineering, Imperial College London, South Kensington Campus, London SW7 2AZ, United Kingdom.
4 Kutateladze Institute of Thermophysics, Novosibirsk, Russian Federation.
5 Department of Energy Sciences, Lund University, P.O. Box 118, SE-22100 Lund, Sweden.
6 Research & Development Institute of Northwestern Polytechnical University in Shenzhen, Shenzhen, 518057, Guangdong, China.

*Corresponding address: School of Marine Science and Technology, Northwestern Polytechnical University, China. E-mail: xgn@nwpu.edu.cn (G. Xie).
†Corresponding address: Clean Energy Processes (CEP) Laboratory, Department of Chemical Engineering, Imperial College London, UK. E-mail: c.markides@imperial.ac.uk (C.N. Markides).

Abstract

In this paper, we consider the flow and heat transfer behaviour of turbulent upward and downward flows of supercritical n-decane, in order to reveal the features of heat transfer deterioration (HTD) that would be expected in relevant active regenerative cooling systems for scramjet engines. Specific focus is placed on key velocity-field features that appear in these flows. Following the validation of six turbulence models, the SST k-ω and RNG k-ε models are found to be suitable for simulating the upward and downward flow cases, respectively. “M” type velocity profiles (a non-monotonicity of the velocity along the radial direction) are observed, which arise due to a spatially-varying interplay between the inertial and viscous forces in the flow domain, while larger velocity gradients in the buffer layer are also observed that contribute to the phenomenon of HTD. Furthermore, it is found that the secondary flows as well as the different mass fluxes that arise due to the velocity increase from the wall to the flow core zone (i.e., the influencing range and intensity of cross-sectional kinetic energy), respectively, are observed in the HTD development region, as well as the HTD peak area and degradation regions. A zone of higher thermal diffusion appears in the near-wall region, which acts as a thermal barrier and contributes to HTD.

Keywords:
heat transfer deterioration; secondary flow; supercritical n-decane; thermal diffusion; velocity strains.
Nomenclature

- \( a \) thermal diffusivity of fluid, \( \lambda/c_p, \text{m}^2/\text{s} \)
- \( A \) area of a cross-sectional, \( \text{m}^2 \)
- \( c_p \) specific heat, \( \text{J/kg} \cdot \text{K} \)
- \( d \) diameter, \( \text{m} \)
- \( D_\omega \) orthogonal divergence term, \( \text{kg/m} \cdot \text{s}^4 \)
- \( g_i \) gravitational volume force along the \( i \) direction, \( \text{m/s}^2 \)
- \( G_b \) turbulent kinetic energy term generated by the buoyancy lift term, \( \text{kg/m} \cdot \text{s}^3 \)
- \( G_k \) turbulent kinetic energy generated by laminar velocity gradient, \( \text{kg/m} \cdot \text{s}^3 \)
- \( G_\omega \) turbulent kinetic energy generated by \( \omega \) equation, \( \text{kg/m} \cdot \text{s}^4 \)
- \( h \) heat transfer coefficient, \( \text{W/m}^2 \cdot \text{K} \)
- \( k \) turbulent kinetic energy, \( \text{m}^2/\text{s}^2 \)
- \( L \) axial or radial boundary
- \( p \) pressure, \( \text{Pa} \)
- \( Pr \) Prandtl number, \( \nu/a \)
- \( Pr_t \) turbulent Prandtl number, \( (\mu_t/\rho)/(\lambda_t/\rho c_p) \)
- \( q_w \) heat flux, \( \text{W/m}^2 \)
- \( q_m \) mass flow rate, \( \text{kg/s} \)
- \( Re \) Reynolds number, \( u_d u/\nu \)
- \( S_k \) turbulent kinetic energy term, \( \text{kg/m} \cdot \text{s}^3 \)
- \( S_c \) turbulent dissipation source term, \( \text{kg/m} \cdot \text{s}^4 \)
- \( t \) temperature, \( ^\circ \text{C} \)
- \( T \) temperature, \( \text{K} \)
- \( u \) local velocity, \( \text{m/s} \)
- \( x \) \( x \)-dimension length, \( \text{mm} \)
- \( y \) \( y \)-dimension length, \( \text{mm} \)
- \( y^+ \) dimensionless wall distance, \( y \rho u_c/\mu \)
- \( Y_k \) turbulence generated by diffusion, \( \text{kg/m} \cdot \text{s}^3 \)
- \( Y_m \) contribution of turbulent pulse expansion to the dissipation rate in the global flow, \( \text{kg/m} \cdot \text{s}^3 \)
- \( Y_\omega \) turbulence generated by diffusion, \( \text{kg/m} \cdot \text{s}^4 \)
- \( z \) \( z \)-dimension length, \( \text{mm} \)
Greek

\( \Gamma_\omega \) diffusion rate of \( \omega \), kg/m·s

\( \varepsilon \) turbulent dissipation rate, m\(^2\)/s\(^3\)

\( \lambda \) thermal conductivity, W/m·K

\( \mu \) dynamic viscosity, kg/m·s

\( \nu \) kinematic viscosity, m\(^2\)/s

\( \rho \) density at the bulk temperature, kg/m\(^3\)

\( \omega \) specific dissipation rate, m\(^2\)/s\(^3\)

Subscripts

\( b \) bulk

\( d \) downstream section

\( i \) local position

\( in \) inlet or inner wall

\( out \) outer wall

\( pc \) pseudo-critical temperature

\( sec \) secondary flows

\( t \) test section

\( u \) upstream section

\( w \) wall
1. INTRODUCTION

Hypersonic vehicles are considered as a new aerial combat platform due to perfect stealth, a high Mach number flight and a wide range of strikes. The scramjet, which uses a hydrocarbon fuel as a propellant, is widely regarded as the core component of hypersonic vehicles [1]. The heat load of the combustor increases sharply due to the action of supersonic combustion and aerodynamic heating. The temperature and heat flux at a Mach number of 8 are 4000 K, 10-20 MW/m², respectively, and the temperature exceeds the limits of existing materials [2]. An active regenerative cooling system can perfectly solve this problem [3-5] and the hydrocarbon fuel, e.g., RP-3 is used as a refrigerant in cooling channels. Generally, the critical pressure of the hydrocarbon fuel, e.g., RP-3, is 2-3 MPa and these always work at supercritical condition due to the operating pressures of 3.0-7.0 MPa of the combustor [6,7]. The thermophysical properties of supercritical hydrocarbon fuels change drastically and abnormal heat transfer phenomena can be observed such as a heat transfer deterioration (HTD) [8-11]. Thus a large number of research works has paid attention to the convection heat transfer characteristics of supercritical hydrocarbon fuels. Because the components of hydrocarbon fuels are numerous and it is not realistic to consider them all in numerical simulations. Thus, surrogate models of hydrocarbon fuels are receiving much more attention [12-14]. In this work, supercritical n-decane is considered to be the refrigerant in the cooling channels of the active regenerative cooling system.

In addition to the studies on supercritical hydrocarbon fuels, supercritical water and carbon dioxide (CO₂) in pipes with different types and orientations, i.e., vertical [15,16], horizontal [17,18] and helically coiled [19,20] pipes have also attracted attention. An abnormal heat transfer phenomenon can be found collectively in the thermal performance of these supercritical fluids. Fu et al. [21] and Jiang et al. [22] experimentally studied the different influence factors on the heat transfer of supercritical RP-3 and carbon dioxide such as system pressure, heat flux, mass flow rate, flow direction, etc. They reported that the sharp variation of the thermophysical properties plays an important role in the deteriorated heat transfer zone. Furthermore, Pioro et al. [23] deemed that the occurrence of the HTD can be accurately evaluated by one correlation. Li et al. [24] used the Shear-Stress Transport (SST) k-ω model to investigate the flow and heat transfer of supercritical water flowing in internally ribbed tubes. They reported that ribbed geometries hardly work but mixed ones play an important role by suppressing the buoyance force. Jackson [25] studied the flow and heat transfer in the presence of a buoyance force in a vertical tube and found that the forced convection turns into free convection with an increase of the buoyancy force. He et al. [26] experimentally researched the heat transfer of supercritical R245fa flowing vertically upward in a circular tube and it was observed that the phenomenon of heat transfer deterioration appeared at moderate heat and
mass fluxes. Liao and Zhao [27] numerically explored laminar convection of CO\textsubscript{2} in a vertical mini/micro tube and it was revealed that the buoyancy effect plays an important role in a small tube even for high Reynolds numbers. Bovard et al. [28] numerically investigated the heat transfer of supercritical CO\textsubscript{2} and water and it was demonstrated that the increase of mass flux and operating pressure can decrease the effect of thermal-induced acceleration and buoyance force. These conclusions are consistent with Ref. [29]. Pucciarelli et al. [30] used LES (Large Eddy Simulation) to analyse the coupling effect of fluid heat transfer and wall heat conduction. It was suggested that the function of wall heat conduction should be considered explicitly. Tao et al. [31] proved that the buoyancy, density fluctuation and variation profoundly influence the accuracy of turbulence models. By modifying the LS (Launder-Sharma) model, the precision was improved by 41% compared to the experimental data. Kline et al. [32] experimentally explored the onset of HTD for CO\textsubscript{2} flowing upward in electrically heated vertical tubes and the minimum heat flux was given. Jaromin and Anglart [33] investigated heat transfer of supercritical water in a vertical tube at deteriorated conditions and they also found that the wall temperature is greatly affected by the Prandtl number.

Through the above-mentioned literature survey, the parametric variation, buoyancy effect, heat transfer correlation, heat transfer of enhanced structures, prediction models of heat transfer deterioration, modifications of the turbulence models on the abnormal heat transfer of supercritical fluids were widely studied. However, the mechanism of heat transfer deterioration is still unclear and this point was confirmed in Ref. [34]. In this context, the motivation here is to study the fundamental mechanism of heat transfer deterioration. It is well known that the effect of the flow field on the temperature field is very strong but a more detailed analysis of the flow field was not mentioned in the open literature. In this paper, the information of flow behaviour is used to illustrate the heat transfer characteristics, since the velocity distribution and flow structure demonstrate the heat transfer intensity in some certain areas, and the velocity fields and velocity gradients are used to indicate the global velocity structures and velocity trends, respectively. The velocity vector is applied to show the flow direction and the effect of secondary flow on the abnormal heat transfer, particularly for heat transfer deterioration, is also explained.

2. COMPUTATIONAL DOMAIN

2.1 Active regenerative cooling

As described above, the active regenerative cooling technique of interest to the present work is regarded as a particularly effective approach for absorbing the combustor-generated heat as shown in Fig. 1. The hydrocarbon fuel firstly flows into channels and the heat in the combustor wall is taken
up by the hydrocarbon fuel. The heated fuel enters the combustion chamber to participate in the combustion and generate thrust. In addition, heated fuel is used as a heat source in the power generation system and this research is being pushed forward. Actually, the supercritical hydrocarbon fuel will be cracked once the temperature reaches a certain value. The heat flux along the combustor wall is non-uniform. However, in this article, the basic law of the abnormal heat transfer phenomenon is only focused on, thus the simple boundary conditions are considered, i.e., uniform heat flux is applied in the non-cracking zone.

In addition, a circular pipe is employed in this study and the reasons are as follows: actually, rectangular channels are always used in the active regenerative cooling system [35,36], but it has not been finally determined based on Ref. [37], and circle, trapezoid, triangle cross-section shapes were also studied in detail. Thus it is worthwhile to investigate heat transfer deterioration in circular pipes. Besides, for aviation aircraft with a turbine, aviation kerosene can be used in a kerosene-air heat exchange system [38] and such information has been mentioned in Ref. [39]. A part of the compressed air flows into the kerosene-air heat exchanger and is cooled by the supercritical aviation kerosene. The high-quality air (low-temperature) is applied to cool the turbine blades. In this cooling process, circular pipes are always utilized. Thus it is worthwhile to investigate heat transfer deterioration in circular pipes. On the other hand, the experimental data in circular pipes can be widely obtained [40,41], but those data in rectangular channels are not readily available. This is another reason why heat transfer deterioration in circular pipes is widely investigated. Overall, a circular pipe is considered and it is heated circumferentially in this article.

2.2 Computational domain

As stated above, a circular pipe is the object of the study. The computational domains for cases of flow upward and flow downward in a circular pipe, respectively, are studied according to the experimental conditions in Ref. [40] and these computational domains can be seen in Fig. 2. The computational domain with a total length of 959 mm includes three parts: an upstream section with a length of 100 mm, a test section with a length of 759 mm and a downstream section with a length of 100 mm. A fully developed flow situation is guaranteed by an upstream section and backflow is avoided by a downstream section. The wall heat conduction has a strong effect on the heat transfer of supercritical hydrocarbon fuel in cooling channels and this view has been verified by Refs. [42,43]. Thus the solid zone (stainless steel) and liquid zone (supercritical n-decane) are both considered in the simulation process. The outside and inside diameters of the circular pipe are 3.02 mm and 2.00 mm, respectively. It should be noted that the heat is only applied to the test section.
3. NUMERICAL METHOD

3.1 Thermophysical properties of n-decane

Before the numerical calculations are conducted, the physical properties of the supercritical fuel should be determined and these data can be calculated by NIST (National Institute of Standards and Technology) [44]. The variations of the thermophysical properties of n-decane with the change of fluid temperature can be found in Refs. [45,46] and are also plotted in Fig. 3. Herein, the function of piecewise-linear in the commercial software Fluent was applied and 30 points were embedded. It should be noted that more data were used in the area of drastic changes of physical properties to ensure the accuracy of the calculations.

3.2 Numerical methods and boundary conditions

The computational domain (solid and fluid zones) is discretized by a structured mesh and this process is completed by the software ICEM CFD 15.0. An equidistant mesh is used in the solid zone, and there is no boundary layer. The O-type mesh is adopted in the fluid zone, and the mesh, which is closest to the walls, is generated densely. It should be noted that the enhanced wall treatment is used to solve the near-wall region in this work and the dimensionless wall distance $y^+$ should be less than unity. To ensure that the value of $y^+$ is less than unity, the number of mesh layers near the wall is set to 30. Moreover, the height of the first mesh layer and growth rate are 0.001 mm and 1.01, respectively. It is well known that the flow and thermal performance of the fluids are controlled by the Navier-Stokes equations and a turbulence model in the numerical computations. These equations are solved by the commercial software FLUENT 15.0. The coupling of pressure and velocity is handled by the method of the SIMPLEC (Semi-Implicit Method for Pressure Linked Equations-Consistent) algorithm. The relaxation factors are taken as the default values at first, and then they are reduced appropriately to achieve convergence. In the module of spatial discretization, the least-squares cell-based and second-order accurate formulas are used to solve the gradient and pressure, respectively. Besides, the second-order upwind scheme is applied to solve the equations of momentum, turbulent kinetic energy, and specific dissipation rate, respectively. The calculations are terminated once the residuals of all equations are less than $10^{-5}$.

The boundary conditions are essential for smooth computations and those in Ref. [40] are borrowed for cases of flow upward and flow downward: inlet mass flow rate $q_m = 1.585$ g/s, inlet temperature $t_{in} = 150 ^\circ$C, operating pressure $p = 3$ MPa, heat flux $q_w = 306$ kW/m², outflow for the outlet. Also, gravity is considered due to the importance of the buoyance force in supercritical heat transfer. These boundary conditions also can be seen in Fig. 2. A hybrid initialization is used to obtain the initial conditions.

3.3 Validation of turbulence model

The differences between existing mature turbulence models lead to a variation of the simulation
accuracy for the same problem. The precise selection of a turbulence model plays a crucial role in the simulation process. Besides, an appropriate model should be an efficient tool to validate the consistency between the computational results and test data. In the present work, some mature turbulence models, including the standard $k$-$\varepsilon$ (2 equations), RNG $k$-$\varepsilon$ (2 equations), realizable $k$-$\varepsilon$ (2 equations), standard $k$-$\omega$ (2 equations), SST $k$-$\omega$ (2 equations), transition SST (4 equations), Reynolds stress (7 equations) are considered as comparative models and used to simulate the thermal behaviours of supercritical n-decane flowing upward and downward in a vertically circular pipe. Accordingly, the experimental data in Ref. [40] are selected to verify the validity and accuracy of the models, and the following experimental conditions are given by $Re_{in} = 4000$, $p_{in} = 3$ MPa, $t_{in} = 150$ ℃, $d_{in} = 2$ mm, $q_w = 306$ kW/m$^2$ (upward flow) and $q_w = 312$ kW/m$^2$ (downward flow). The wall temperature along the flow direction is regarded as a parameter of evaluation for all turbulence models and relevant numerical results are exhibited in Fig. 4. It is shown that the wall temperature predicted by the different models is strongly affected by the flow direction in the circular pipe.

For the flowing upward case, it is found that the prediction deviation by the transition SST (4 equations) is larger than that by the other models and thus this turbulence model is excluded. The results predicted by the standard $k$-$\varepsilon$ (2 equations), RNG $k$-$\varepsilon$ (2 equations) and realizable $k$-$\varepsilon$ (2 equations) are almost same, and the numerical and experimental data in the range of $x/d_{in} = 175$–320 match well with a maximum deviation of 10%. However, the HTD phenomenon is not captured by these three turbulence models and thus they are not considered for the present simulations. The $k$-$\omega$ two-equation model can capture the HTD phenomenon relatively well. The region of the occurrence of HTD is closer to that of in the test condition whilst the deviation is relatively large. Accordingly, the standard $k$-$\omega$ model is not considered. The results calculated by the Reynolds stress model (7 equations) match the experimental data well in the range of $x/d_{in} = 165$–365 and the maximum deviation is 10%. However, the deviation in the entrance region is larger and the HTD location is further delayed. Thus the Reynolds stress model in this paper is not either considered.

On the other hand, the results calculated by the SST $k$-$\omega$ (2 equations) turbulence model match well with the experimental data in the initial stage of heat transfer. Although there are some deviations (the maximum deviation is 16%) in the later stage, the HTD phenomenon can be captured distinctly. The reason for the decrease in prediction accuracy when the temperature increases is explained as follows: the n-decane will undergo a cracking reaction once the temperature exceeds 770 K and the properties of n-decane will be changed sharply, as confirmed by reference [47]. However, the cracking reaction is not considered in this study and the prediction accuracy of numerical simulation is reduced once the temperature of the hydrocarbon fuel increases. In a conclusion, the SST $k$-$\omega$ turbulence model is applied to calculate the flow and thermal performance of the flowing upward case.
For the flowing downward case, the RNG $k$-$\varepsilon$ model in the downward flow has a lower deviation than the SST $k$-$\omega$ model and other turbulence models. Thus the RNG $k$-$\varepsilon$ model is applied to calculate the flow characteristics and thermal performance of the flowing downward case.

### 3.4 Mesh independence

A proper balance between precision and CPU demands is a goal in pursuing the numerical simulations. This strongly depends on the selection of the mesh number. In this work, the following simulation conditions for the upward-flowing case in a vertically circular pipe are applied: $Re_{in} = 4000$, $p_{in} = 3$ MPa, $d_{in} = 2$ mm, $d_{out} = 3.02$ mm, $t_{in} = 150$ °C, $q_{w} = 306$ kW/m$^2$. Figure 5(a) exhibits a structured mesh of O-type and the mesh numbers are decided by the boundary layer and the number of radial and axial nodes. The details of the axial and radial node settings are shown in Table 1. To ensure that the value of $y^*$ is less than unity, the number of mesh layers near the wall is set to 30. Moreover, the height of the first mesh layer and growth rate are 0.001 mm and 1.01, respectively. The mesh settings in the solid zone and fluid zone are both verified and the outlet fluid temperature is about 816 K for all cases. Besides, the mesh numbers of 4.0M, 5.0M, 5.8M and 6.5M are selected for further evaluation, respectively. The local inner wall temperature is regarded as the parameter and target of the validation, and relevant numerical results are displayed in Fig. 5(b). It can be observed that the wall temperature has a relatively small deviation when the mesh number is equal or above 5.8M. Hence, the following simulations adopt the mesh number of 5.8M for all cases. Moreover, the values of $y^*$ are 0.07-1.04, 0.07-1.03 for the cases of flow upward and flow downward, respectively.

### 3.5 Governing equations

The flow and heat transfer processes of supercritical hydrocarbon fuel flow examined in this work are governed by the mass, and energy conservation equations, which are given as follows.

**Mass conservation equation:**

$$\frac{\partial \hat{\rho} \hat{u}_i}{\partial x_i} = 0 \quad (1)$$

**Momentum conservation equation:**

$$\frac{\partial \hat{p} \hat{u}_i \hat{u}_j}{\partial x_j} = -\frac{\partial \hat{\rho} \hat{u}_i' \hat{u}_j''}{\partial x_j} - \frac{\partial \hat{\rho}}{\partial x_i} + \frac{\partial (2\mu \delta_{ij})}{\partial x_j}$$

$$+ \frac{\partial \mu}{\partial x_i} \left[ \hat{u}_i (\hat{r}_{ij} - \hat{\rho} u_i' u_j'') \right] \quad (2)$$

**Energy conservation equation:**

$$\frac{\partial \hat{p} \hat{u}_i \hat{H}}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \hat{q}_j - \hat{\rho} u_i' h'' + \hat{r}_{ij} u_i'' - \rho u_i' \frac{1}{2} u_i'' u_i'' \right]$$

$$+ \frac{\partial}{\partial x_j} \left[ \hat{u}_i (\hat{r}_{ij} - \hat{\rho} u_i' u_j'') \right] \quad (3)$$

It is noted that there is a term of the turbulent stress $-\hat{\rho} u_i' u_j''$ in the Favre-averaging momentum conservation equation (Eq. 2). Herein, the eddy viscosity model is used and a parameter
of the turbulent viscosity (or eddy viscosity) is introduced to express the turbulent stress as a function of the turbulent viscosity by using the Boussinesq hypothesis.

To determine the turbulent viscosity, there are some established eddy viscosity models including the zero equation model, one equation model and two equations model. According to the above validation, the SST $k$-$\omega$ turbulence model and RNG $k$-$\varepsilon$ turbulence model are applied in this study, and the corresponding equations are given below.

**Transport equation of SST $k$-$\omega$:**

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i} \right] + G_k - Y_k + S_k$$  \hspace{1cm} (4)

$$\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_i \omega)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \Gamma_w \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + D_\omega + S_\omega$$  \hspace{1cm} (5)

where $k$ is the turbulent kinetic energy, $\omega$ is the specific dissipation, $G_k$ is the turbulence kinetic energy generated by laminar velocity gradient, $G_\omega$ is generated by the $\omega$ equation, $\Gamma_w$ is diffusion rates $\omega$, $Y_k$ and $Y_\omega$ are turbulence terms generated by diffusion, $D_\omega$ is the orthogonal divergence term and $S_k$ and $S_\omega$ are user-defined source terms.

**Transport equation of RNG $k$-$\varepsilon$:**

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k$$  \hspace{1cm} (6)

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \alpha_k \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_3 \varepsilon G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon$$  \hspace{1cm} (7)

where $k$ is the turbulent kinetic energy, $\varepsilon$ is the turbulent dissipation rate, $G_k$ is the turbulence kinetic energy generated by laminar velocity gradient, $G_b$ is the turbulent kinetic energy generated by buoyancy lift, $Y_m$ is the contribution of turbulent pulse expansion to the dissipation rate in the global flow, $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_3$ are constants, and $S_k$ and $S_\varepsilon$ are turbulent kinetic energy and turbulent dissipation source terms, respectively.

### 4. RESULTS AND DISCUSSION

#### 4.1 Heat transfer partitions

Figure 6 shows how the inner wall temperature is changing along the flow direction for the upward-flowing and downward-flowing cases. There is a large difference between the two cases in terms of the inner wall temperature and this means that the heat transfer behaviour is inconsistent for different flow directions. For the upward-flowing case, an HTD (a typical form of heat transfer) can be observed.
Generally, the heat transfer can be separated in the following regions: inlet region, normal heat transfer region, HTD development region, HTD degradation region, heat transfer recovery region and outlet region. On the other hand for downward flow, the phenomenon of HTD does not show up. However, the inner wall temperature at the inlet region is larger than that for the upward-flow case. To explore the differences of heat transfer mechanisms between the upward-flowing case and downward-flowing case, three positions ($z = 450$ mm, $489$ mm, $530$ mm) are selected and these positions are located in the HTD development region, HTD peak and HTD degradation region, respectively. In this paper, the phenomenon of HTD is the main research objective and more attention is paid to that. The following sections will focus on HTD and the relevant characteristics and mechanisms will be described.

Also, the wall heat transfer coefficients along the $z$-axis in the vertical pipe are determined: (a) flow upward case and (b) flow downward case are depicted in Fig. 7 and its expression is given below.

$$h = \frac{q_w}{T_w - T_{in}}$$  \hfill (8)

where $q_w$ is the heat flux, W/m$^2$, $T_w$ is the temperature of the interface between the heated wall and fluid zone, K, inlet temperature $T_{in}$ is regarded as a reference value, K. The change of heat transfer coefficient agrees well with the change of the inner wall temperature. For the flow upward case, an obvious decrease of the heat transfer coefficient is found and this means that an HTD phenomenon occurs. For the flow downward case, the heat transfer coefficient decreases gently except for a sharp decrease in the initial stage.

Figure 8 displays the fluid temperature at $y = 0$ for the upward-flowing and downward-flowing cases. A distinct difference in the temperature profiles is found between the upward-flowing case and the downward-flowing case. Also, the temperature gradient is not consistent between these two cases. To quantitatively analyze the influence of the temperature gradient on the HTD phenomenon, the temperature gradients along the $x$-axis at positions $z = 450$ mm, $z = 489$ mm, and $z = 530$ mm for (A) Upward-flowing case and (B) Downward-flowing case, respectively, are evaluated as shown in Fig. 9. For the upward-flowing case, the temperature gradient near the wall firstly increases and then decreases to zero. The maximum value is about $2.8 \times 10^3$ K/mm. For the downward-flowing case, the temperature gradient in the vicinity of the wall drops rapidly to zero. The maximum value is about $7.5 \times 10^3$ K/mm. Obviously, the temperature gradient near the wall in the upward-flowing case is smaller than that in the downward-flowing case. It can be seen from Fig. 6 that the wall temperatures at positions $z = 450$ mm, $z = 489$ mm, and $z = 530$ mm in the upward-flowing case are larger than those in the downward-flowing case. However, the bulk temperature is almost the same for these two cases as shown in Fig. 9. Based on the equation
where $\lambda$ is the thermal conductivity of the fluid, $\Delta t$ is the temperature difference between the wall temperature and fluid temperature, it is obvious that the heat transfer coefficient in the upward-flowing case is lower than that in the downward-flowing case. The contributions to the different temperature gradients and the appearance of HTD phenomenon are of interest to reveal. This question should be considered based on more analysis.

4.2 Velocity field

It is known that the flow and heat transfer affect and restrict each other, thus the velocity field can be considered to discuss the HTD behaviour. Figure 10 shows the velocity fields along the $z$-axis on the streamwise-spanwise plane ($y = 0$) for the upward-flowing case and the downward-flowing case, respectively. Large differences can be found obviously between the upward-flowing and downward-flowing cases. In the upward flows, an “M” type velocity profile is presented at $z = 450-650$ mm. As is well-known, the velocity profile is related to the density and dynamic viscosity, as well as the turbulence state. Herein, the density and dynamic viscosity are used to illustrate the formation of the velocity profile.

It is found from Fig. 3 that the density and dynamic viscosity of n-decane are reduced dramatically with the increase of the temperature in the vicinity of the pseudo-critical point. It is noted that the pseudo-critical temperature ($T_{pc} = 648.2$ K) at $p_{pc} = 3$ MP appears at $z = 450-650$ mm. At the same cross-section, the values of density and dynamic viscosity near the wall are smaller than those in the flow core zone. Based on the mass conservation, the velocity increases with the decrease of density. Furthermore, the forward flow resistance is also reduced due to the decrease of dynamic viscosity. Thus the velocity near the wall (excluding the boundary layer (no-slip boundary effects)) is larger than that in the flow core zone. This means that a drastic variation in density and dynamic viscosity leads to a change in the flow pattern. Such a velocity profile reduces the shear stress and hence the turbulent kinetic energy. In other words, the inertial force works effectively in the region near the wall. However, a “U” type velocity profile is found in the same region ($z = 450-650$ mm) for the downward-flowing case. Obviously, the velocity near the wall is not larger than that in the flow core zone. In other words, the viscous force still plays a major role in controlling the velocity.

The relationship between momentum inertia and viscosity can be described by the Reynolds number, which represents a measure of the ratio between inertial and viscous forces as follows:

$$Re_i = \frac{u_i d_{in}}{v_i} = \frac{\rho_i u_i d_{in}}{\mu_i} = \frac{q_m d_{in}}{A \mu_i}$$

where $u_i$ is the local fluid velocity, $d_{in}$ is the hydraulic diameter, i.e., inside diameter, $v_i$ is the local
kinematic viscosity, $\rho_i$ is the local fluid density, $\mu_i$ is the local dynamic viscosity, $q_m$ is the local mass flow rate, $A$ is the cross-sectional area. Figure 11 shows the distribution of the Reynolds number on the streamwise-spanwise plane ($y = 0$), upward and downward. It can be seen that the Reynolds number is larger for the upward-flowing cases than for the downward-flowing cases in the region of $z = 450$-$650$ mm. As stated above, the inertial force dominates the flow condition in the region of large Reynolds number such as the upward-flowing case, while the viscous force dominates the flow condition in the region of small Reynolds number such as the downward-flowing case. It is noted that the difference in Reynolds number is small after $z = 650$ mm, thus the colour of the legend looks the same. Actually, the magnitude of the Reynolds number at $z = 650$-$859$ mm is not uniform because of the change of thermophysical properties.

4.3 Variation of velocity strains

In Section 4.2, the velocity field is introduced to describe the characteristics of upward-flowing and downward-flowing cases at $y = 0$. The condition of HTD is determined by the relative magnitude of inertia forces and viscous forces. In this section, the velocity gradient at three positions ($z = 450$ mm, 489 mm, 530 mm) is depicted to further illustrate the flow features of the upward- and downward-flowing cases. Figure 12 shows the gradient of the $z$-velocity along the negative $x$-axis direction at different boundary layers for the upward-flowing and downward-flowing cases. The boundary layer consists of the viscous sublayer, buffer layer, and fully turbulent region and the definition of these boundary layers can be found in our previous research [48]. There is a relation between the normal distance from the wall ($x$-axis in this study) and the dimensionless distance from the wall ($y^+$). Thus the normal distance from the wall can be used to determine the value of $y^+$. Then the different boundary layers can be ensured based on the dimensionless distance from the wall. At the position $z = 450$ mm, the velocity increases from the wall to the flow core zone. In the buffer layer, the velocity gradient in the upward flow is larger than that in the downward flow. In the fully turbulent region, the magnitude of the two cases is the opposite. At the position $z = 489$ mm, the relative magnitude of the velocity along the $x$-axis is not unique. Similarly, the velocity gradient in the buffer layer is larger than that in the fully turbulent region but the relative magnitude becomes large. Furthermore, the equilibrium point moves to the centreline in the fully turbulent region. At the position $z = 530$ mm, the magnitude of the velocity along the $x$-axis tends to increase gradually. The situation in the buffer layer between the two cases is similar but the relative magnitude decreases. The equilibrium point moves closer toward the centreline. Through the above analysis, the important result is summarized as follows: a large velocity gradient in the buffer layer and the non-monotonicity of velocity along the radial direction contribute to the phenomenon of HTD.
4.4 Velocity vector

Figure 13 shows distributions of the velocity vector at different cross-sections (z = 450 mm, 489 mm, 530 mm) for the upward-flowing and downward-flowing cases. It is worth noting that the length of lines and the size of arrows represent the magnitude of velocity. The flow direction can be seen directly in Fig. 13. For the upward-flowing and downward-flowing cases, respectively, the opposite flow between the wall area and core zone is gradually formed. At position z = 450 mm, either the upward-flowing case or downward-flowing case, the secondary flow points to the central region but the magnitude of the velocity is different. This means that the direction of secondary flow does not contribute to the development of HTD but the magnitude of the secondary flow does. At position z = 489 mm, the flow direction is separated for the two cases but the location of the separation is different. In the upward flow, the location of flow separation is farther away from the wall compared to the downward flow. This implies that the extent of the dead zone is large for the upward flow and even though the magnitudes of the velocity towards the wall and to the core zone, respectively, are consistent, it still cannot change the situation of HTD. At position z = 530 mm, similarly, the extent of the dead zone is large for the upward flow but the magnitude of the velocity towards the wall is also large. This indicates that more fluid mixes and interacts in the dead zone. This feature inhibits the phenomenon of HTD.

According to the above analysis, it can be revealed that most fluid flows into the core zone and this behaviour causes the phenomenon of HTD in the development region. In the peak area of HTD, the dead zone near the wall contributes to the behaviour of HTD and further aggravates this feature at a balanced state for the fluid flowing into the wall and core zone. In the degradation region, more fluid flows into the wall and this action effectively inhibits HTD.

4.5 Evolution of secondary flow

According to Ref. [49], the cross-section kinetic energy can be used to quantitatively analyse the evolution characteristics of the secondary flow and it is expressed as follows:

\[ k_{sec} = \frac{1}{A} \int_A \left( u_x^2 + u_y^2 \right) dA \]  

where \( k_{sec} \) is the cross-section kinetic energy, \( A \) is the area of the cross-section, \( u_x \) and \( u_y \) are the flow velocities along the x-axis and y-axis, respectively. Figure 14 shows the cross-section kinetic energy on the streamwise-spanwise plane (y = 0). It is found that the cross-section kinetic energy in the region \( z = 400-700 \text{ mm} \) in the upward-flowing case is larger than that in the downward-flowing case, which suggests that the secondary flow is indeed the cause of HTD. Below, combined with the position of pseudo-critical temperature as shown in Fig. 15, the active mechanisms of the secondary flow in both upward and downward flows are explained in detail below.
Upward-flowing case:

For the development region of HTD (\(z = 400-489\) mm), the influence of secondary flow moves from the region near the wall to the flow core zone. Correspondingly, it can be observed from Fig. 15 that the positions of the pseudo-critical temperatures are 0.82 mm, 0.70 mm away from the centre at the positions \(z = 450\) mm, 489 mm, respectively. These positions are almost coinciding with the positions of maximum cross-section kinetic energy. It means that the secondary flow does affect the HTD development and the heat transfer deteriorates severely as the pseudo-critical temperature moves toward the flow core zone.

For the degradation region of HTD (\(z = 489-700\) mm), the influence range and magnitude of the secondary flow become larger than that in the development region of HTD. This affects both the regions near the wall and in the flow core zone. This feature inhibits the further development of HTD. It implies that the complete development of the secondary flow is beneficial for the reduction of HTD. In brief, the influencing mechanism of HTD in view of secondary flow is different for the development region of HTD and degradation region of HTD, respectively, in the upward-flowing case. The secondary flow can not only aggravate the HTD but also weakens the HTD. It depends on the influence range and magnitude of the secondary flow. This result is supported by the findings in Ref. [50] and the buoyancy force was used to determine HTD and HTE (heat transfer enhancement).

Downward-flowing case:

It can be observed from Fig. 15, that the positions of pseudo-critical temperature at \(z = 450\) mm and \(z = 489\) mm, respectively, are inconsistent with the active regions of the secondary flow. Subsequently, the influence area of the secondary flow is far away from the wall or located in both regions near the wall and flow core zone. Thus, the significant HTD is not observed on this occasion.

4.6 Momentum and thermal energy assessment

The Prandtl number is a relative measure of molecular momentum and thermal diffusion. In addition, the ratio of the flow to the thermal boundary layer can also be expressed by the Prandtl number:

\[
Pr_i = \frac{\nu_i}{a_i} = \frac{\mu_i c_{pi}}{\lambda_i}
\]  

(12)

where \(\nu_i\) is the local kinematic viscosity of the fluid, \(a_i\) is the local thermal diffusivity of fluid, \(\mu_i\) is the local dynamic viscosity, \(c_{pi}\) is the local specific heat, \(\lambda_i\) is the local thermal conductivity.

Figure 16 shows the distribution of the Prandtl number at positions \(z = 450\) mm, 489 mm, 530 mm for the upward-flowing and downward-flowing cases. The obvious differences can be found between these two cases. In the flow core zone, the Prandtl number in upward flows is larger than that in the flowing downward case, while the state is the opposite in the region near the wall. Furthermore, the gradient of the Prandtl number is even more pronounced in the upward flow. Herein, Fig. 15 (a) is used...
to discuss the development of HTD. As the fluid moves forward, the value of the Prandtl number increases gradually along the radius from the wall to the core zone. In view of momentum and heat transfer, it means that the capacity of momentum diffusion is increasingly dominant, correspondingly, the capacity of heat diffusion decreases. In view of the boundary layer, the relatively large Prandtl number means that the thermal boundary layer is thin in the region near the wall. This fact is supported by our previous work in Ref. [29]. This indicates that the thermal resistance prevents the heat transfer from the region near the wall to the flow core zone. In other words, once the HTD occurs, the region of larger heat diffusion is located in the area near the wall but it forms a thermal barrier.

5. CONCLUSIONS

In order to cool effectively the combustion chambers of scramjets, turbulent flows of supercritical n-decane are used to absorb the heat in cooling channels. Abnormal flow and heat transfer behaviour is always observed in such flows due to the dramatic variation of thermophysical properties of supercritical fuels, particularly near the pseudo-critical point. To understand the physical mechanisms of unconventional flow and heat transfer, the SST $k-\omega$ and RNG $k-\varepsilon$ turbulence models were used to simulate the turbulent flow and thermal performance of supercritical n-decane in both upward and downward flows in a vertical pipe. Key characteristics of the velocity field such as the Reynolds number, velocity gradients, secondary flows and the Prandtl number were investigated.

The following conclusions can be drawn from the above research:

1. Significantly different results are found for the upward-flowing and downward-flowing cases. HTD is more likely to occur in upward flows, while the inlet region for the downward-flowing case is stronger than that in the upward-flowing equivalent. In the upward-flowing case, an “M” type velocity profile is caused by strong variations of density and dynamic viscosity in the region of HTD, whereas a “U” type velocity profile is found in the downward flows. This feature can be explained as follows: the inertial force plays a major role in the region near the wall for the upward-flowing case while the viscous force is important for the downward-flowing case.

2. The $z$-velocity gradient component along the negative $x$ direction at position $z = 489$ mm for the upward- and downward-flowing cases was used to explain the observed HTD by comparing to those at positions $z = 450$ mm and $z = 530$ mm. Furthermore, the characteristics of the velocity gradient in the buffer layer and fully turbulent region, respectively, were also discussed. At position $z = 489$ mm, in the buffer layer, the velocity gradient of the upward-flowing case is larger than that in downward-flowing case. In the fully turbulent region, backflow or a vortex can be found. Thus it is concluded that a large velocity gradient in the buffer layer and the non-monotonicity of velocity...
(backflow or vortex) along the radial direction contribute to the phenomenon of HTD.

3. In the upward-flowing case, the development region of the HTD, the degradation region of HTD and the peak area of HTD depend on the position and intensity of the secondary flow. In the development region, HTD arises due to fluid flowing into the core zone. In the peak area, the dead zone near the wall contributes to HTD, and in the degradation region, the action of fluid flowing into the wall effectively inhibits HTD. Furthermore, the secondary flow was used to analyse the differences in HTD behaviour in the development and degradation regions for the upward-flowing case. The secondary flow can either aggravate or weaken the HTD phenomenon. In the downward-flowing case, the HTD phenomenon is not observed due to the influence area of the secondary flow, which is far away from the wall or located in both regions near the wall and flow core zone.

4. In the flow core zone, the Prandtl number in an upward flow is larger than that in the downward-flowing case, which reverses in the region near the wall. In the upward-flowing case, this leads to a larger thermal boundary layer and a higher thermal resistance that prevents heat from the region near the wall to transfer to the flow core zone. Furthermore, considering the velocity field at the position at which HTD occurs, a region of higher heat diffusion is located in the area near the wall, which forms and acts as a thermal barrier.

ACKNOWLEDGMENTS

This work was sponsored by the National Natural Science Foundation of China (51676163), the National 111 Project (Grant No. B18041), the Fundamental Research Funds of Shenzhen City (JCYJ20170306155153048), and the Innovation Foundation for Doctor Dissertation of Northwestern Polytechnical University (Grant No. CX202029), and was financially supported by the China Scholarship Council (CSC) who gave Mr. Y. Li the opportunity to perform part of his PhD studies at Lund University. The work was also supported by Russian Government “Megagrant” project 075-15-2019-1888.
REFERENCES


[38] Bruening GB, Chang WS. Cooled cooling air systems for turbine thermal management. *International Gas Turbine and Aeroengine Congress and Exhibition*, 1999, Jun 7-10, Indianapolis, Indiana, USA.


## Tables

**Table 1.** The details of the axial and radial node settings

<table>
<thead>
<tr>
<th>Radial direction</th>
<th>Axial direction</th>
<th>Mesh size (nodes)</th>
<th>Fluid outlet temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L_1$</td>
<td>$L_2$</td>
<td>$L_3$</td>
<td>$L_4$</td>
</tr>
<tr>
<td>15</td>
<td>15</td>
<td>21</td>
<td>10</td>
</tr>
<tr>
<td>15</td>
<td>15</td>
<td>21</td>
<td>15</td>
</tr>
<tr>
<td></td>
<td></td>
<td>25</td>
<td>50</td>
</tr>
<tr>
<td>15</td>
<td>15</td>
<td>30</td>
<td>15</td>
</tr>
<tr>
<td>17</td>
<td>17</td>
<td>30</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>19</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Fig. 1: Schematic diagram of the active regenerative cooling system: the scramjet combustor with cooling channels; physical heat absorption and chemical heat absorption, non-uniform heat flux and typical service environment at supercritical pressure operation.
Fig. 2: Computational domains for cases of flow upward and flow downward.
Fig. 3: Variation of thermophysical properties of n-decane with the change of temperature.
Fig. 4: Validation of turbulence models for the upward-flowing case and downward-flowing case by comparing present numerical results and experimental data in Ref. [40]: (a) flow upward with $q_w = 306$ kW/m$^2$, and (b) flow downward with $q_w = 312$ kW/m$^2$. 
Fig. 5: Mesh distribution and independence validation: (a) structured mesh used in the axial cross-section; the O-type is used in the fluid zone and the initial mesh height and the growth rate is 0.001 mm and 1.01, respectively; equidistant mesh is used in the solid zone, and (b) local inner wall temperature along the z-axis for different mesh numbers: 4.0M, 5.0M, 5.8M, 6.5M, and the same conditions: $d_{in} = 2$ mm, $d_{out} = 3.02$ mm, $p = 3$ MPa, $q_{w} = 306$ kW/m², $Re_{in} = 4000$, $t_{in} = 150$ °C in a vertical circular pipe.
Fig. 6: Inner wall temperature (circumferential mean) along the $z$-axis in a vertical pipe: (a) flow upward case: development region of HTD, HTD peak, degradation region of HTD, and (b) flow downward case: no HTD.
Fig. 7: Wall heat transfer coefficient along the \( z \)-axis in a vertical pipe: (a) flow upward case, and (b) flow downward case.
Fig. 8: Distribution of the fluid temperature on the streamwise-spanwise plane ($y = 0$): the upward-flowing case with large temperature gradient near the wall at an HTD region, and the downward-flowing case with small temperature gradient near the wall at the same positions.
Fig. 9: Temperature gradients and temperature distribution along the x-axis at the positions $z = 450$ mm, $z = 489$ mm, and $z = 530$ mm for (A) Upward-flowing case and (B) Downward-flowing case.
Fig. 10: Velocity field along $z$-axis on the streamwise-spanwise plane ($y = 0$): the upward-flowing case with M-type velocity profile at an HTD region and the downward-flowing case with U-type velocity profile at the same positions.
Fig. 11: Distribution of Reynolds number on the streamwise-spanwise plane ($y = 0$); the upward-flowing case with large Reynolds number at HTD region, and the downward-flowing case with relatively small Reynolds number at the same positions.
Fig. 12: Gradient of z-velocity along the negative x-axis direction at different boundary layer locations (viscous sublayer, buffer layer and fully turbulent region) for the upward-flowing case and flowing downward case: (a) z = 450 mm, (b) z = 489 mm, and (c) z = 530 mm.
Fig. 13: Distribution of velocity vector (x-velocity and y-velocity) at different cross-sections; upward flow: (a) $z = 450$ mm, (b) $z = 489$ mm, (c) $z = 530$ mm; downward flow: (d) $z = 450$ mm, (e) $z = 489$ mm, and (f) $z = 530$ mm.
Fig. 14: Exhibition of cross-section kinetic energy on the streamwise-spanwise plane ($y = 0$): the upward-flowing case with relatively large cross-section kinetic energy at $z = 400-700$ mm, and the downward-flowing case with relatively small cross-section kinetic energy at the same positions.
Fig. 15: Change of temperature along the positive radius direction at different cross-sections ($z = 450$ mm, 489 mm, 530 mm); the position of pseudo-critical temperature from the core zone in three cross-sections can be found for the upward-flowing case and downward-flowing case.
Fig. 16: Prandtl number at different cross-sections ($z = 450$ mm, 489 mm, 530 mm): (a) the upward-flowing case with relatively small Prandtl number near the wall, and (b) the downward-flowing case with relatively large Prandtl number near the wall.